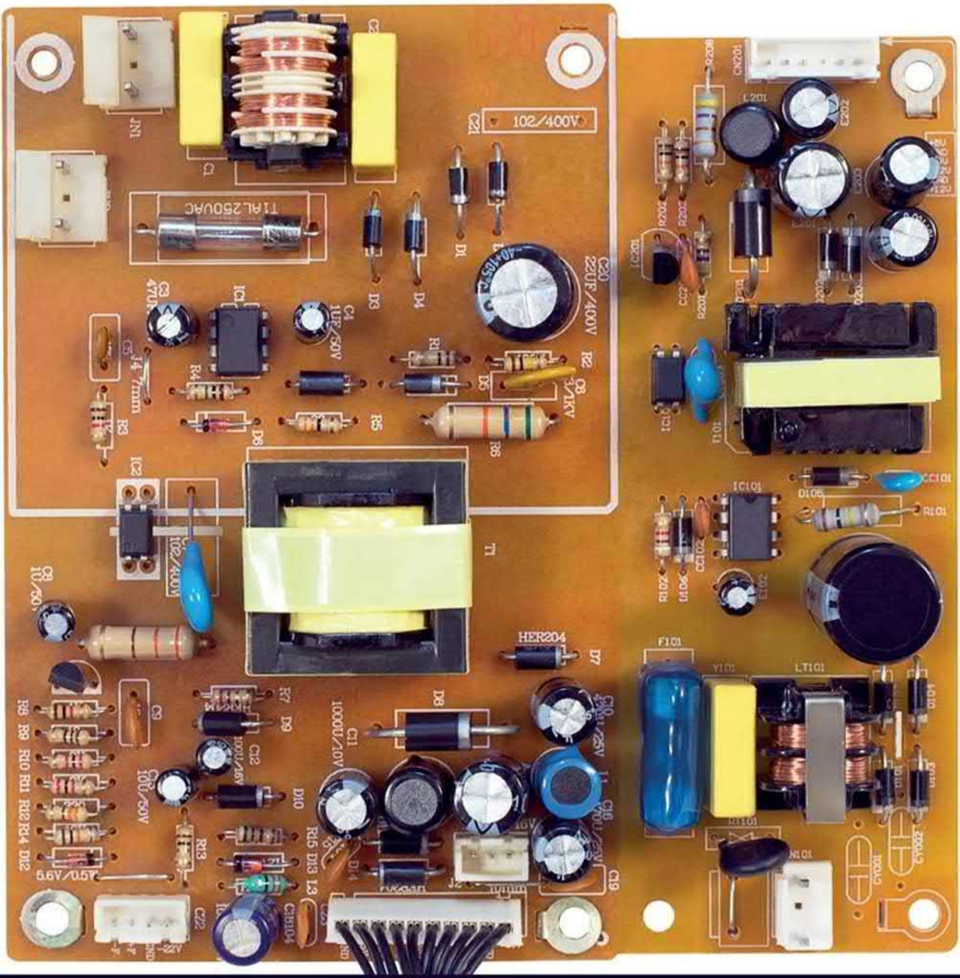


SIMULATION OF POWER ELECTRONICS CONVERTERS USING PLECS®

FARZIN ASADI
KEI EGUCHI

VISIT…

LANZAROTE

Caliente.com

Simulation of Power Electronics Converters Using PLECS®

Simulation of Power Electronics Converters Using PLECS®

Farzin Asadi

Kei Eguchi

ELSEVIER

ACADEMIC PRESS
An imprint of Elsevier

Academic Press is an imprint of Elsevier
125 London Wall, London EC2Y 5AS, United Kingdom
525 B Street, Suite 1650, San Diego, CA 92101, United States
50 Hampshire Street, 5th Floor, Cambridge, MA 02139, United States
The Boulevard, Langford Lane, Kidlington, Oxford OX5 1GB, United Kingdom

© 2020 Elsevier Inc. All rights reserved.

No part of this publication may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, recording, or any information storage and retrieval system, without permission in writing from the publisher. Details on how to seek permission, further information about the Publisher's permissions policies and our arrangements with organizations such as the Copyright Clearance Center and the Copyright Licensing Agency, can be found at our website: www.elsevier.com/permissions.

This book and the individual contributions contained in it are protected under copyright by the Publisher (other than as may be noted herein).

Notices
Knowledge and best practice in this field are constantly changing. As new research and experience broaden our understanding, changes in research methods, professional practices, or medical treatment may become necessary.

Practitioners and researchers must always rely on their own experience and knowledge in evaluating and using any information, methods, compounds, or experiments described herein. In using such information or methods they should be mindful of their own safety and the safety of others, including parties for whom they have a professional responsibility.

To the fullest extent of the law, neither the Publisher nor the authors, contributors, or editors, assume any liability for any injury and/or damage to persons or property as a matter of products liability, negligence or otherwise, or from any use or operation of any methods, products, instructions, or ideas contained in the material herein.

Library of Congress Cataloging-in-Publication Data
A catalog record for this book is available from the Library of Congress

British Library Cataloguing-in-Publication Data
A catalogue record for this book is available from the British Library

ISBN 978-0-12-817364-0

For information on all Academic Press publications
visit our website at https://www.elsevier.com/books-and-journals

Publisher: Brian Romer
Acquisition Editor: Lisa Reading
Editorial Project Manager: Andrae Akeh
Production Project Manager: Sruthi Satheesh
Cover Designer: Matthew Limbert

Typeset by SPi Global, India

ELSEVIER Book Aid International
Working together
to grow libraries in
developing countries
www.elsevier.com • www.bookaid.org

Dedication

In loving memory of my mother, Khorshid Tahmasebi (1950-2019), always on my mind, forever in my heart.

Farzin Asadi

Dedicated to my lovely family.

Kei Eguchi

Preface

Power electronic systems are widely used today to provide power processing for applications ranging from computing and communications to medical electronics, appliance control, transportation, and high-power transmission. The associated power levels range from milliwatts to megawatts.

Modeling and simulation are essential ingredients of the analysis and design process in power electronics. They help a design engineer gain an increased understanding of circuit operation. With this knowledge the designer can, for a given set of specifications, choose a topology, select appropriate circuit component types and values, estimate circuit performance, and complete the design by ensuring that the circuit performance will meet specifications even with the anticipated variations in operating conditions and circuit component values.

Power electronics systems are nonlinear variable structure systems. They involve passive components such as resistors, capacitors, and inductors, semiconductor switches such as thyristors and MOSFETs, and circuits for control. The analysis and design of such systems presents significant challenges. Fortunately, increased availability of powerful computer and simulation programs makes the analysis/design process much easier.

PLECS® (Piecewise Linear Electrical Circuit Simulation) is one of the modern circuit simulators developed by Plexim® (www.plexim.com). Using PLECS, simulation of power electronics converters can be done easily in a user-friendly environment. PLECS has a rich component library. Its library contains electrical components (resistors, inductors, capacitors, semiconductor switches, OP-AMPs, electrical machines, etc.), magnetic components (magnetic cores, air gap, winding, mmf source, etc.), thermal components (heat sink, thermal resistor/capacitor, thermometer, etc.), mechanical components (force/speed/position sensor, mass, spring, mechanical loads, etc.). PLECS simulation runs quite fast, as well. PLECS has another nice aspect: the trial version of PLECS can be used free of charge for period of 1 month.

PLECS comes in two versions: standalone and Simulink® versions. Standalone version uses its own solver and it can run independently. Simulink version (as the name suggests) runs under the Simulink program and uses the Simulink solver. The Simulink version has a big advantage: when you use the Simulink version of PLECS, you can use all the Simulink blocks in your simulations. This can simplify the simulation considerably. For example, when you want to simulate a power electronics converter, which uses fuzzy logic controller, there is no need to implement the controller block from ground up. You simply use the ready-to-use fuzzy logic controller block of Simulink.

This book tries to show you how simple you can simulate the power electronics converter circuits in PLECS environment. The prerequisite for this book is a first course on power electronics. The studied examples are selected among the most basic circuits of power electronics. All the details are shown, so you can follow the examples easily. It is highly recommended to do some hand calculations for the given examples and compare the results with the one produced by PLECS. Try to find the source of discrepancy if hand analysis and simulation results are not the same. This helps you to learn the concepts deeply. For instance, the voltage drop of diodes is neglected in hand analysis. So, the hand analysis result (which ignores the voltage drop of diodes) and simulation result (which considers the voltage drop of diodes) are not the same.

We want to acknowledge the Plexim for providing the access to PLECS during the writing of this book. We hope that this book will be useful to the readers, and we welcome comments on the book. Enjoy the world of PLECS!

Farzin Asadi
Kei Eguchi

Chapter 1

Brief introduction to PLECS

Chapter Outline

Introduction 1
1.1 What is PLECS? 1
1.2 What is this book? 3
References 3
Further reading 3

Introduction

This chapter gives you a brief introduction about the PLECS and this book.

1.1 What is PLECS?

PLECS (Piecewise Linear Electrical Circuit Simulation) is a software tool for system-level simulation of electrical circuits developed by Plexim [1].

Working with PLECS is quite easy. It has a powerful component library to let you simulate almost every circuit. Your simulation can contain electrical blocks (for instance, semiconductor switch), thermal blocks (for instance, heat sink), magnetic blocks (for instance, a magnetic core), control blocks (for instance, PI controller) and mechanical blocks (for instance, gearbox). Fig. 1.1 shows the available libraries inside the PLECS. We will study these libraries in future chapters. PLECS do the simulations quite fast as well.

PLECS comes in two versions:

- Standalone version,
- Simulink® version.

As the name suggests, the standalone version works independently. You can run it without MATLAB®/Simulink. Simulink version of PLECS uses the Simulink solver as its solver, while the standalone version uses GNU Octave as its numerical engine in place of Simulink. When you work with the Simulink version, all the MATLAB/Simulink blocks are accessible to you. So, you have the power of Simulink behind you.

Plexim provides a free trial version of PLECS, which can be downloaded from https://www.plexim.com/trial. You can use it for a period of 30 days (Fig. 1.2).

Simulation of Power Electronics Converters Using PLECS®. https://doi.org/10.1016/B978-0-12-817364-0.00001-1
© 2020 Elsevier Inc. All rights reserved.

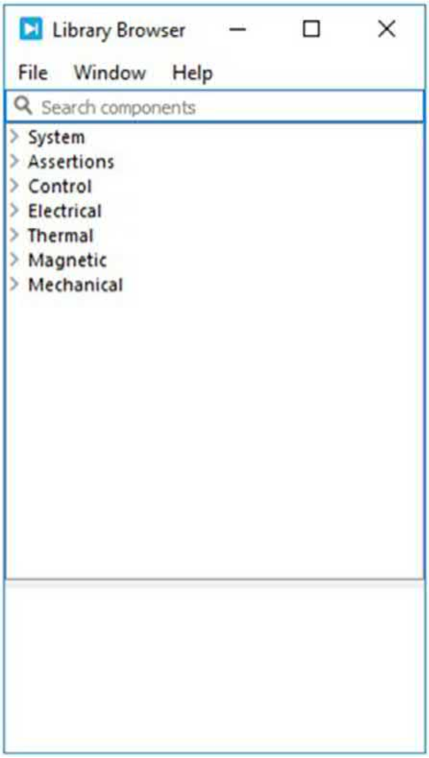
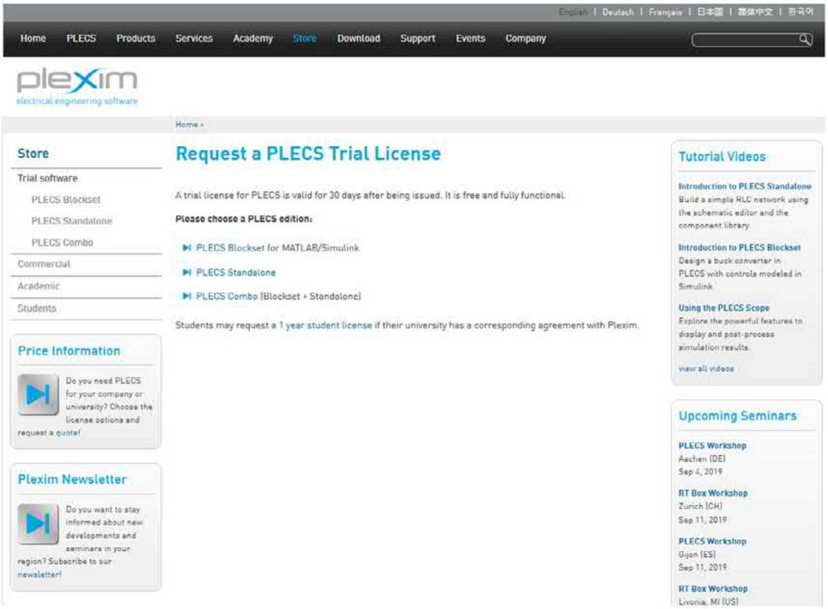

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Thermal
Magnetic
Mechanical

FIG. 1.1 PLECS Library Browser.


English | Deutsch | Français | 日本語 | 簡体中文 | 한국어
Home
PLECS
Products
Services
Academy
Store
Download
Support
Events
Company
plexim
electrical engineering software
Home »
Store
Trial software
PLECS Blockset
PLECS Standalone
PLECS Combo
Commercial
Academic
Students

Request a PLECS Trial License

A trial license for PLECS is valid for 30 days after being issued. It is free and fully functional.

Please choose a PLECS edition:

- PLECS Blockset for MATLAB/Simulink
- PLECS Standalone
- PLECS Combo (Blockset + Standalone)

Students may request a 1 year student license if their university has a corresponding agreement with Plexim.

Price Information

Do you need PLECS for your company or university? Choose the license options and request a quote!

Plexim Newsletter

Do you want to stay informed about new developments and seminars in your region? Subscribe to our newsletter!

Tutorial Videos

Introduction to PLECS Standalone
Build a simple RLC network using the schematic editor and the component library.

Introduction to PLECS Blockset
Design a buck converter in PLECS with controls modeled in Simulink.

Using the PLECS Scope
Explore the powerful features to display and post-process simulation results.

view all videos

Upcoming Seminars

PLECS Workshop
Aachen (DE)
Sep 4, 2019

RT Box Workshop
Zurich (CH)
Sep 11, 2019

PLECS Workshop
Gijon (ES)
Sep 11, 2019

RT Box Workshop
Livonia, MI (US)

FIG. 1.2 https://www.plexim.com/trial.

1.2 What is this book?

This book tries to teach you how simply you can simulate a power electronics converter in PLECS. We strongly suggest you to turn on your computer and regenerate the book examples. Here is a quick review of future chapters:

Chapter 2 shows you the basic skills required to simulate any type of circuits in PLECS. This chapter makes the basis for Chapter 3.

Chapter 3 studies simulation of different power electronics converters. It is a good idea to do some pencil-and-paper analysis for the examples given in this chapter and compare the results with the ones given by PLECS.

Chapter 4 does not gives you fish but teaches you fishing! We suggest you some ways to learn more about the world of circuit simulation using PLECS.

References

[1] PLECS user manual available online at, https://www.plexim.com/download/documentation.

Further reading

[2] Allmeling, J.H., Hammer, W.P. (July 27, 1999). PLECS—piece-wise linear electrical circuit simulation for Simulink. Proceedings of the IEEE 1999 International Conference on Power Electronics and Drive Systems. Vol. 1: 355–360.

Chapter 2

Basics of circuit simulation with PLECS

Chapter outline

Introduction 5
2.1 Example 2.1: Resistive voltage divider 5
  2.1.1 Preparing the simulation 6
  2.1.2 Adding title to the scope 21
  2.1.3 Setting the axis limits 22
  2.1.4 Change the properties of the shown waveform 24
  2.1.5 Reading the values using cursors 24
  2.1.6 Zoom in/out 26
  2.1.7 Exporting the scope block waveforms 29
  2.1.8 Exporting the drawn schematic 31
  2.1.9 Display block 35
  2.1.10 Changing the block names 35
  2.1.11 Hiding the block names 35
  2.1.12 Adding text to the schematic 35
  2.1.13 Ammeter block 37
  2.1.14 Wire colors 38
2.2 Example 2.2: RC circuit analysis 43
  2.2.1 Preparing the simulation 43
  2.2.2 Specifying the initial condition 48
  2.2.3 Showing two or more waveforms simultaneously on the same axis 50
  2.2.4 Multiple input scope 54
  2.2.5 XY scope block 59
  2.2.6 Simulation of control systems 66
  2.2.7 Getting help in PLECS 72
References 74
Further reading 74

Introduction

This chapter studies the fundamental skills one must have when using PLECS standalone to simulate circuits. We show the required techniques using simple circuits such as resistive voltage divider and first-order RC circuit. No power converter is studied in this chapter. Power converter circuits are studied in Chapter 3.

2.1 Example 2.1: Resistive voltage divider

In this example, we analyze the simple circuit shown in Fig. 2.1. We want to find the resistor R2's voltage and the current drawn from the 10-V source. Using basic circuit theory, one can find the answer easily,

Simulation of Power Electronics Converters Using PLECS®. https://doi.org/10.1016/B978-0-12-817364-0.00002-3
© 2020 Elsevier Inc. All rights reserved.

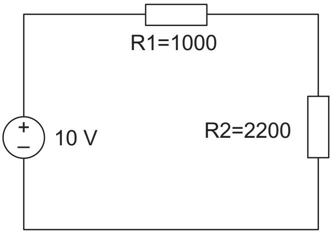

R1=1000
+
−
10 V
R2=2200

FIG. 2.1 Schematic of example 1.

$V_{R2} = \frac{2200}{2200+1000} \times 10\,\text{V} = 6.87\,\text{V}$ and $I = \frac{10}{1000+2200} = 3.12\,\text{mA}$. We want to obtain these results using PLECS.

2.1.1 Preparing the simulation

After running the standalone version of PLECS, windows shown in Figs. 2.2 and 2.3 appear. We use the window shown in Fig. 2.3 to select the blocks that are required to the simulation.

In order to open a new model, click on "New model" as shown in Fig. 2.4. You can make a new model by clicking the File menu and selecting the "New model" in the Library Browser window as well.

PLECS open a new window for you. Circuit diagrams are drawn in this window (Fig. 2.5).

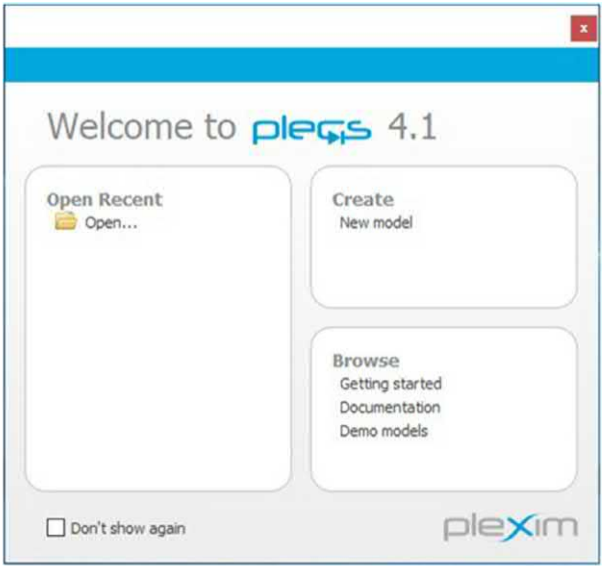

x
Welcome to plecs 4.1
Open Recent
Open...
Create
New model
Browse
Getting started
Documentation
Demo models
☐ Don't show again
plexim

FIG. 2.2 PLECS start page.

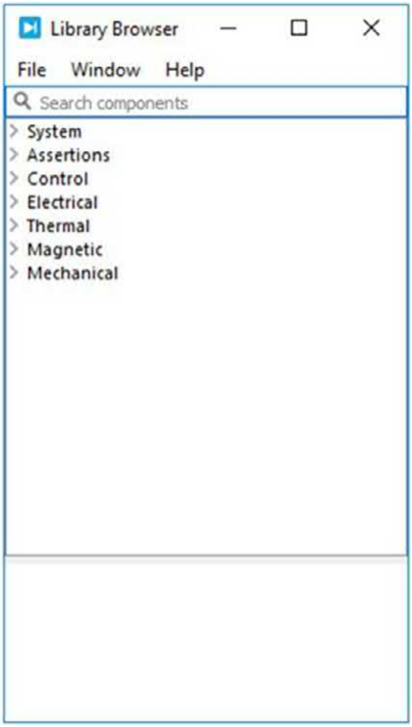

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Thermal
Magnetic
Mechanical

FIG. 2.3 PLECS Library Browser.

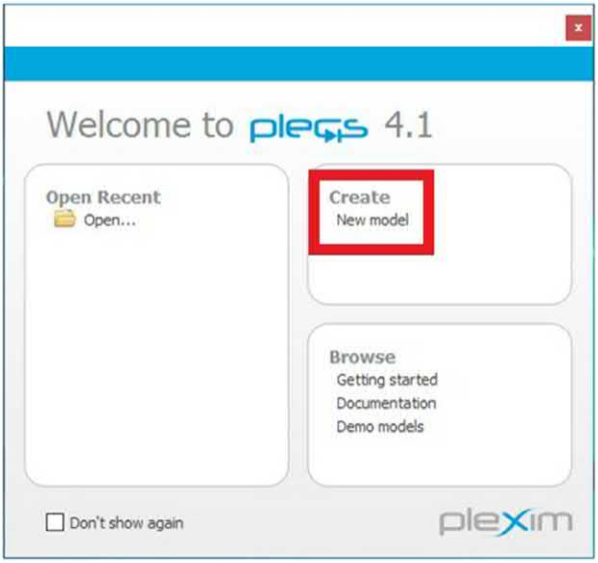

Welcome to plecs 4.1
Open Recent
Open...
Create
New model
Browse
Getting started
Documentation
Demo models
☐ Don't show again
plexim

FIG. 2.4 Creating a new model.

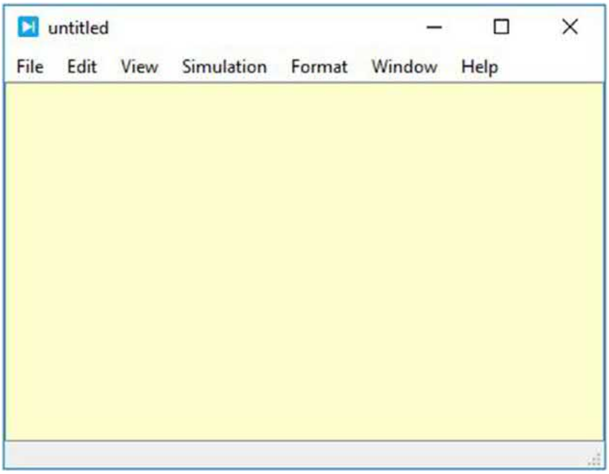


FIG. 2.5 Schematic editor window.

It is a good practice to save the simulation file before starting drawing the schematic (Fig. 2.6). To do this, use the File > Save menu to save the file.

We use the name of firstCircuit for this simulation. PLECS save the files with .plecs extension (Fig. 2.7).

Use the Library Browser window to drag and drop the block named Voltage Source DC to the working area (Fig. 2.8).

After drag and drop, working area look likes that shown in Fig. 2.9.

Use the Library Browser to add a resistor to the schematic (Figs. 2.10 and 2.11).

You can rotate the resistor R1 by clicking on it and pressing the Ctrl+R on your keyboard (Fig. 2.12).

As another method, you can rotate R1 by right clicking on it selecting the Rotate (Fig. 2.13).

Add another resistor to the schematic (Fig. 2.14).

Next step is connecting the parts together. When you bring the mouse pointer near to the parts terminals, mouse pointer changes to a "+" sign. You push down the mouse left button at the source terminal and release it at the desired destination terminal (Fig. 2.15).

We can now set the components values. To do this, double click on the component and enter the desired value. When you check the small box behind the input text boxes, PLECS shows the value entered to that text box in the schematic. For example, if we check the small box behind the Voltage: box (see Fig. 2.16), we will see the schematic shown in Fig. 2.17.

We set the values of remaining parts (Figs. 2.18 and 2.19). After setting the components values, schematic looks like that shown in Fig. 2.20.

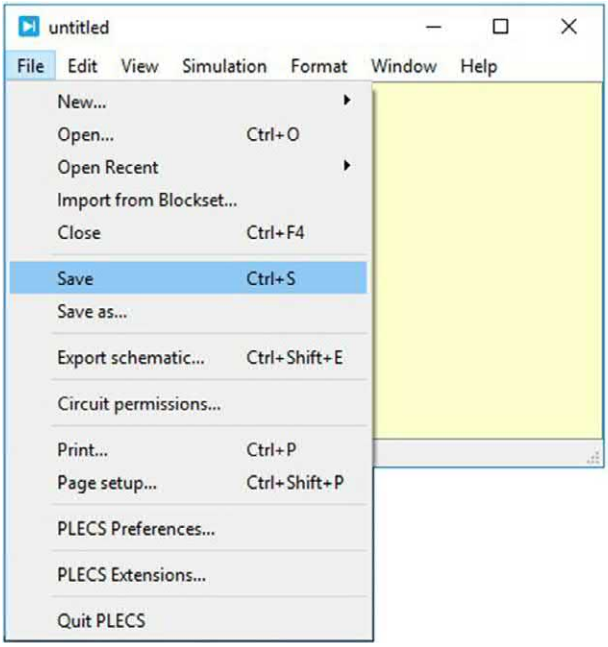
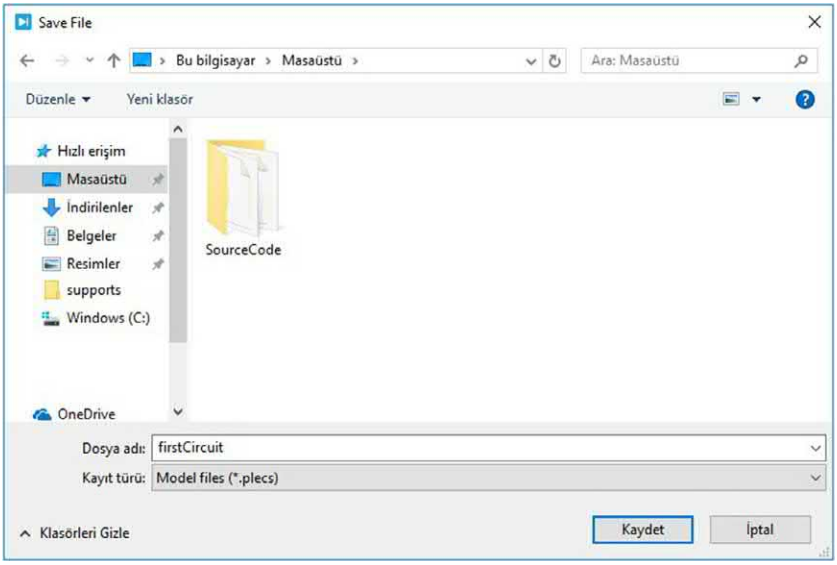
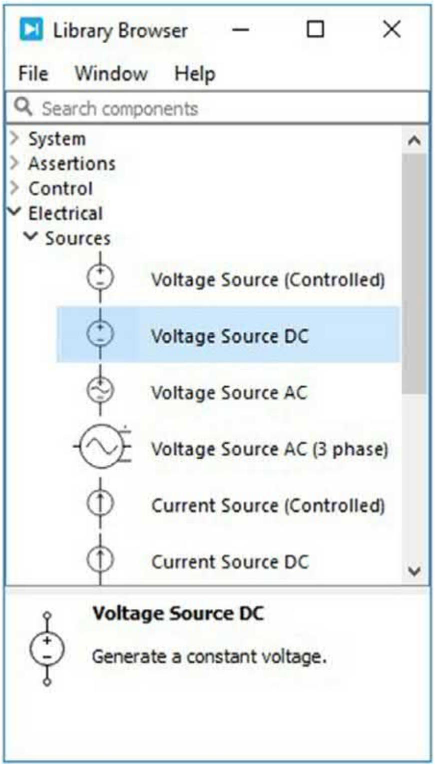

untitled
File
Edit
View
Simulation
Format
Window
Help
New...
Open... Ctrl+O
Open Recent
Import from Blockset...
Close Ctrl+F4
Save Ctrl+S
Save as...
Export schematic... Ctrl+Shift+E
Circuit permissions...
Print... Ctrl+P
Page setup... Ctrl+Shift+P
PLECS Preferences...
PLECS Extensions...
Quit PLECS

FIG. 2.6 Saving the schematic.


Save File
Bu bilgisayar
Masaüstü
Ara: Masaüstü
Düzenle
Yeni klasör
Hızlı erişim
Masaüstü
İndirilenler
Belgeler
Resimler
supports
Windows (C:)
OneDrive
SourceCode
Klasörleri Gizle
Kaydet
İptal

Dosya adı: firstCircuit
Kayıt türü: Model files (*.plecs)

FIG. 2.7 Specifying a path for saving the simulation file.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Voltage Source (Controlled)
Voltage Source DC
Voltage Source AC
Voltage Source AC (3 phase)
Current Source (Controlled)
Current Source DC

Voltage Source DC

Generate a constant voltage.

FIG. 2.8 DC voltage source block.

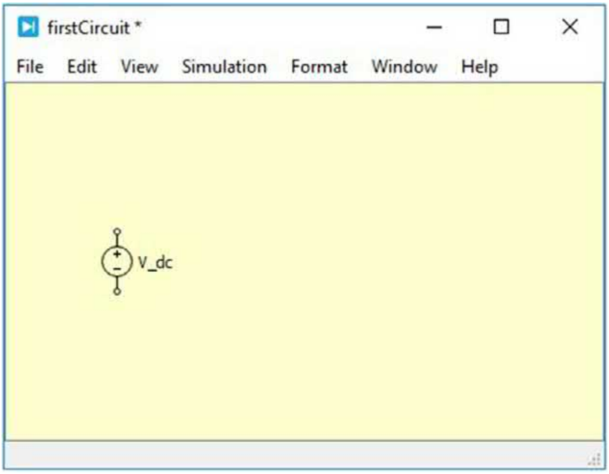

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
V_dc

FIG. 2.9 Placing the voltage source to the schematic.

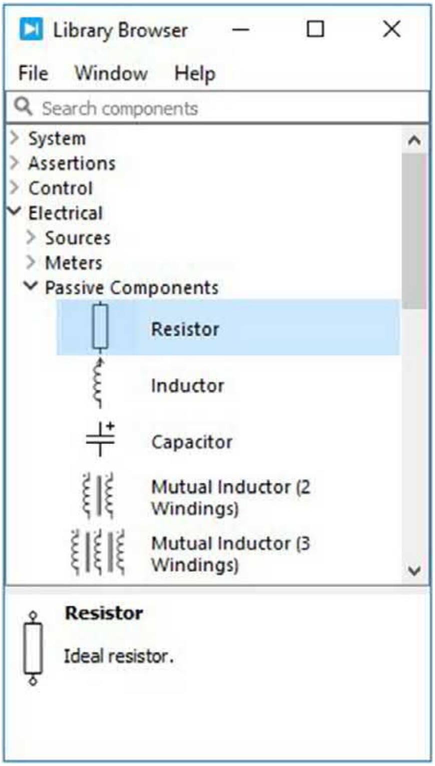

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Resistor
Inductor
Capacitor
Mutual Inductor (2 Windings)
Mutual Inductor (3 Windings)
Resistor
Ideal resistor.

FIG. 2.10 Resistor block.

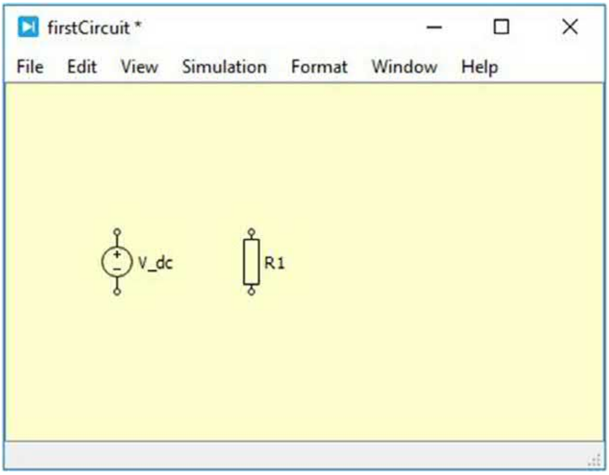

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
V_dc
R1

FIG. 2.11 Placing the resistor block on the schematic.

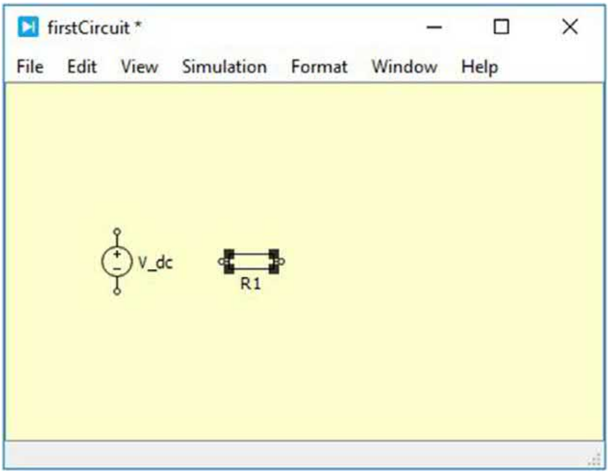

firstCircuit *
File Edit View Simulation Format Window Help
V_dc
R1

FIG. 2.12 Rotating the resistor.

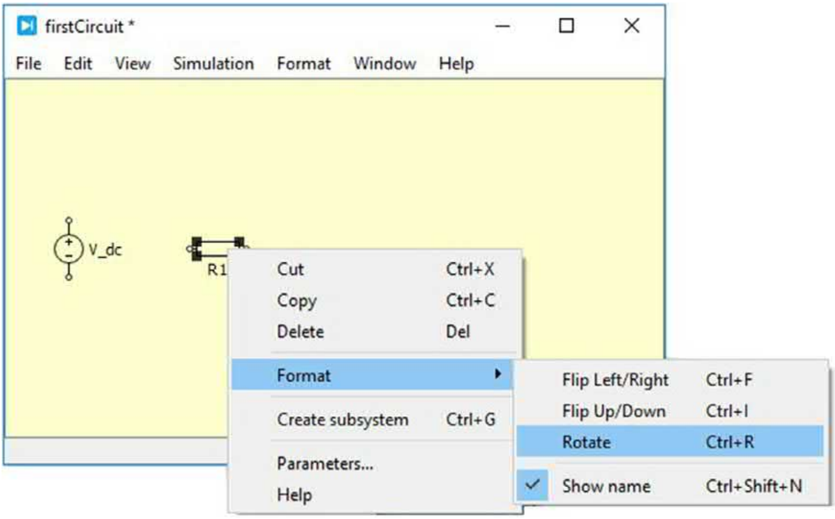

firstCircuit *
File Edit View Simulation Format Window Help
V_dc
R1
Cut Ctrl+X
Copy Ctrl+C
Delete Del
Format
Create subsystem Ctrl+G
Parameters...
Help
Flip Left/Right Ctrl+F
Flip Up/Down Ctrl+I
Rotate Ctrl+R
✓ Show name Ctrl+Shift+N

FIG. 2.13 Rotating the resistor.

In order to measure the output voltage, we must add a voltmeter to the schematic (Figs. 2.21 and 2.22).

Output of voltmeter must be connected to a display to see the waveform. We use a scope block to observe the resistor R2 voltage (Figs. 2.23 and 2.24).

You can ground the circuit by adding a "Electrical Ground" block to your schematic. It is a good practice to use the Electrical Ground block in all of your simulations. "Electrical Ground" can be found in the system menu (Fig. 2.25).

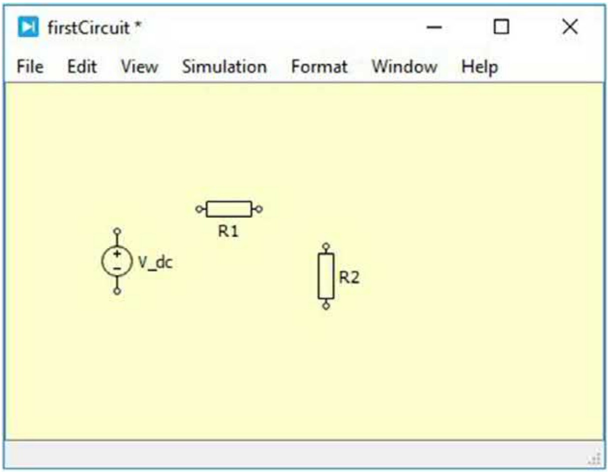

firstCircuit *
File Edit View Simulation Format Window Help
R1
V_dc
R2

FIG. 2.14 Placing the resistor R2 on the schematic editor.

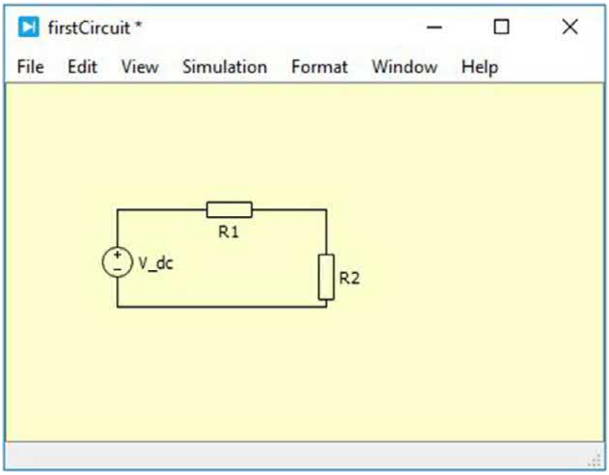
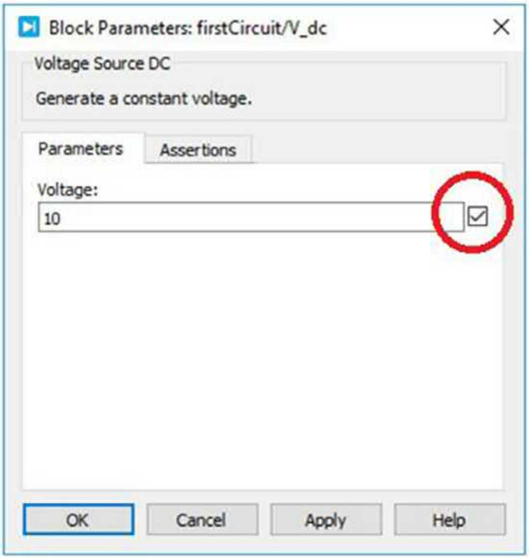

firstCircuit *
File Edit View Simulation Format Window Help
R1
V_dc
R2

FIG. 2.15 Connecting the parts together.

Assume that you forgot the place of "Electrical Ground" block. PLECS contains hundreds of blocks and it is not possible to memorize each blocks place. You can use the "Search components" box of Library Browser. Just type name of the block you need. For example, if you forgot the "Electrical Ground," place just type "ground" in the "Search components" box and press the Enter key on your keyboard (Fig. 2.26).

After adding the "Electrical Ground," schematic looks like that shown in Fig. 2.27.



Block Parameters: firstCircuit/V_dc

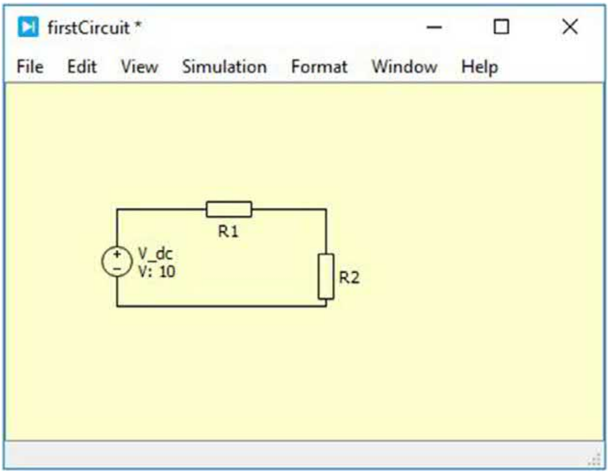
Voltage Source DC

Generate a constant voltage.

Voltage: 10 ☑

Parameters
Assertions
OK
Cancel
Apply
Help

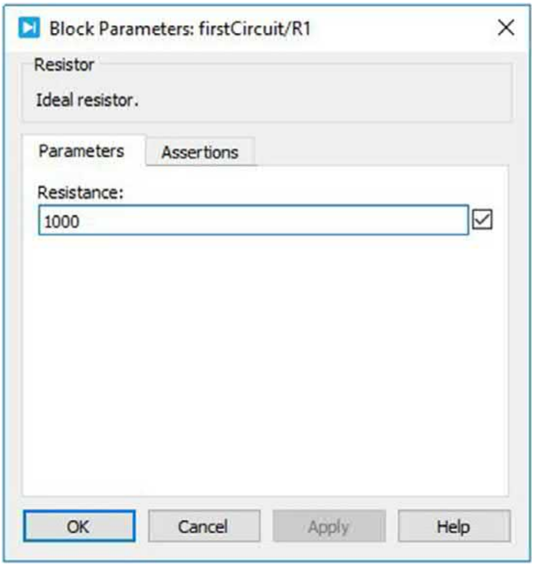
FIG. 2.16 If you check the small box behind the textboxes, PLECS shows that value on the schematic.



firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
R1
V_dc
V: 10
R2

FIG. 2.17 PLECS shows the DC voltage source value (10 V) on the schematic.

Our schematic is ready for simulation now. Before simulating the schematic, we must do some settings (Fig. 2.28). For instance we must tell PLECS to use which type of solver, how accurate results must be, simulation length, etc. These types of settings are done with the aid of “Simulation parameters…”



Block Parameters: firstCircuit/R1

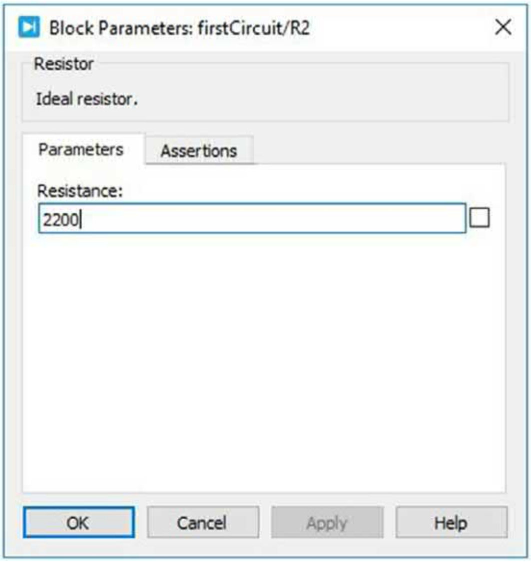
Resistor

Ideal resistor.

Resistance: 1000 ☑

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 2.18 Specifying the value of resistor R1.



Block Parameters: firstCircuit/R2

Resistor

Ideal resistor.

Resistance: 2200 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 2.19 Specifying the value of resistor R2.

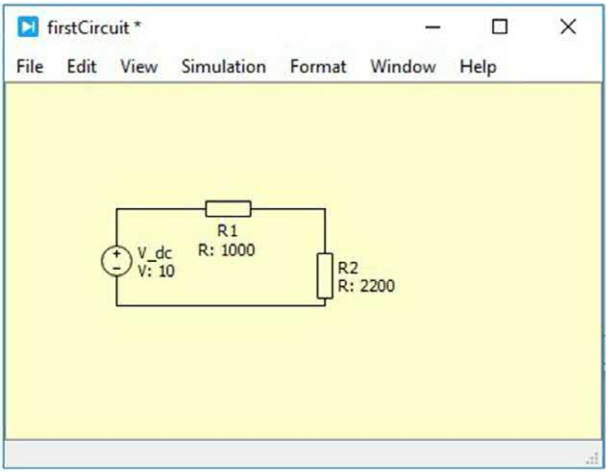
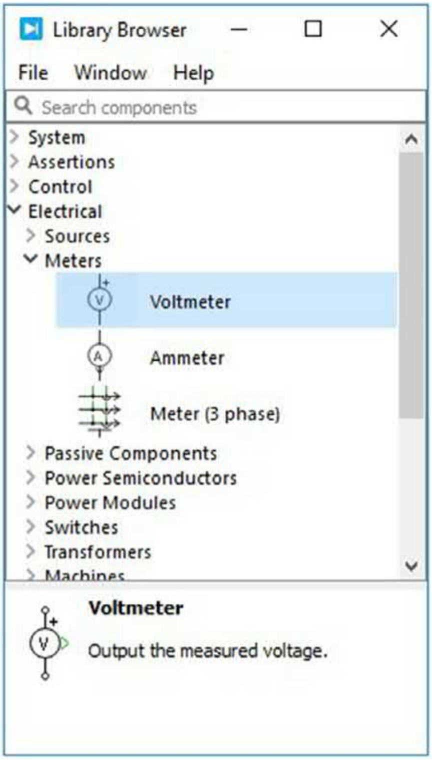

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 1000
V_dc
V: 10
R2
R: 2200

FIG. 2.20 Schematic editor after specifying the parameter values.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Voltmeter
Ammeter
Meter (3 phase)
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines

Voltmeter

Output the measured voltage.

FIG. 2.21 Voltmeter block.

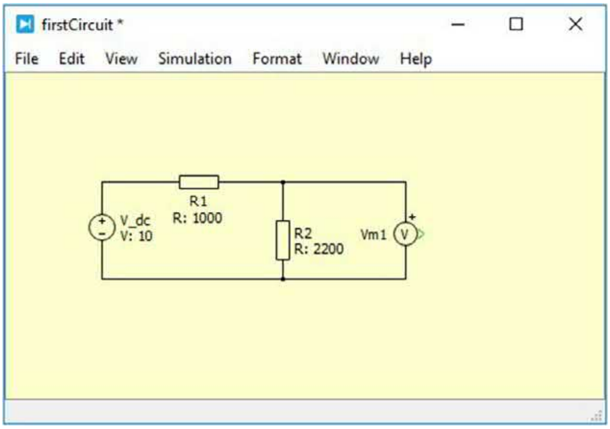
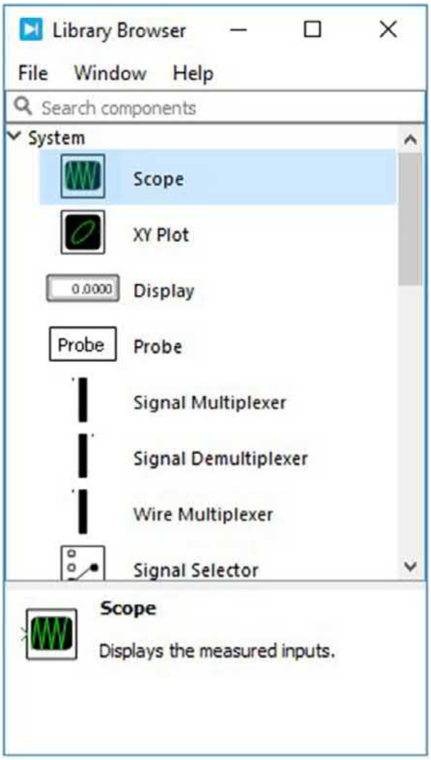

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 1000
V_dc
V: 10
R2
R: 2200
Vm1
V

FIG. 2.22 Connecting the voltmeter to the circuit.


Library Browser
File
Window
Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
Scope

Displays the measured inputs.

FIG. 2.23 Scope block.

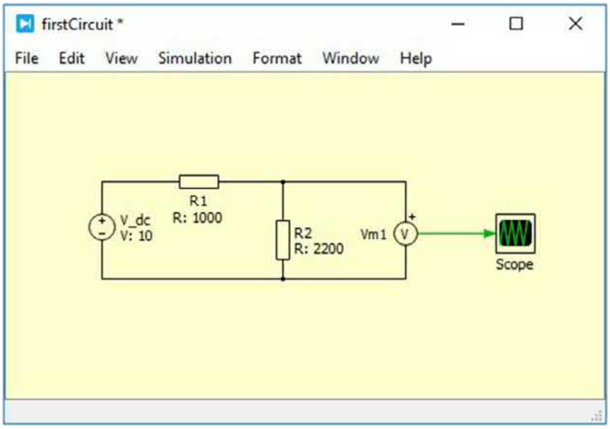
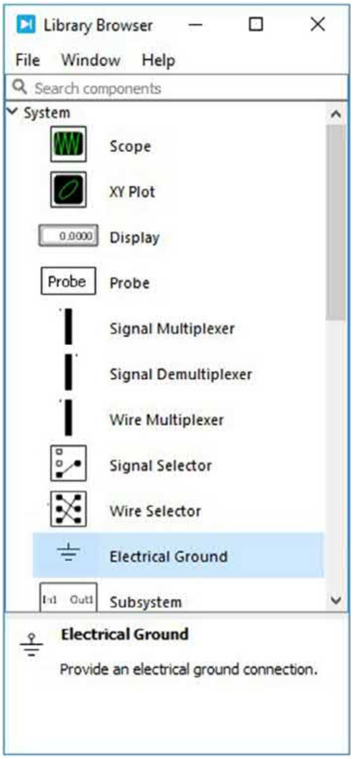

firstCircuit *
File Edit View Simulation Format Window Help
R1
R: 1000
V_dc
V: 10
R2
R: 2200
Vm1
V
Scope

FIG. 2.24 Connecting the scope block to the output of voltmeter.


Library Browser
File Window Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
Wire Selector
Electrical Ground
In1 Out1
Subsystem
Electrical Ground

Provide an electrical ground connection.

FIG. 2.25 Electrical Ground block.

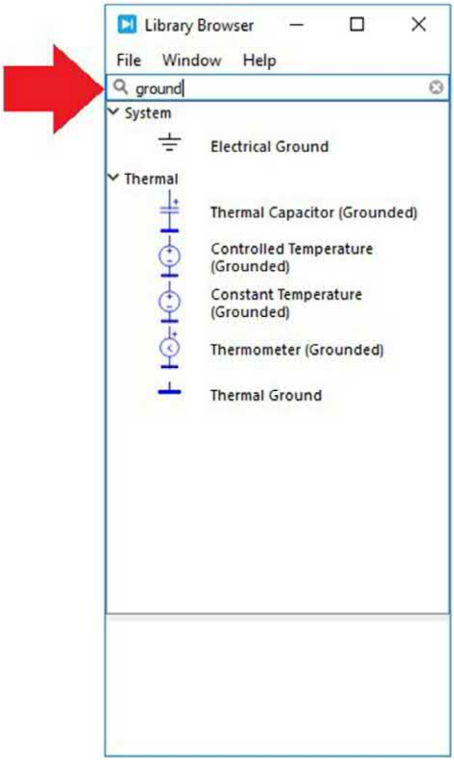

Library Browser
File
Window
Help
ground
System
Electrical Ground
Thermal
Thermal Capacitor (Grounded)
Controlled Temperature (Grounded)
Constant Temperature (Grounded)
Thermometer (Grounded)
Thermal Ground

FIG. 2.26 Searching for block.

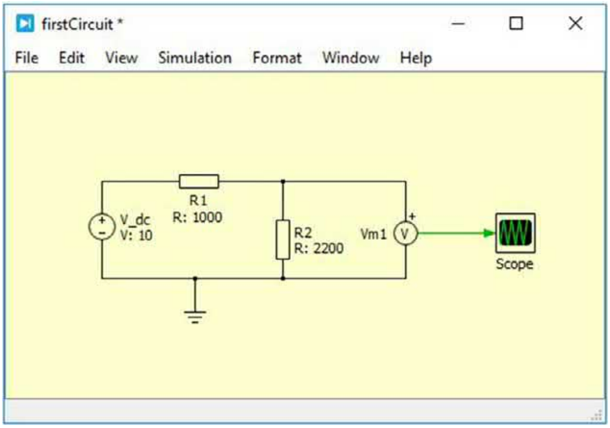

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 1000
V_dc
V: 10
R2
R: 2200
Vm1
V
Scope

FIG. 2.27 Grounding the circuit.

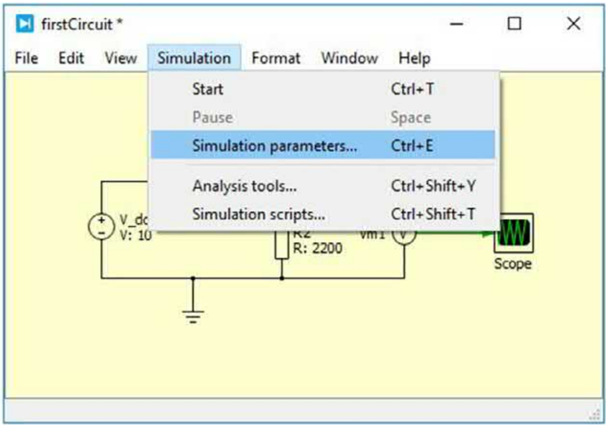
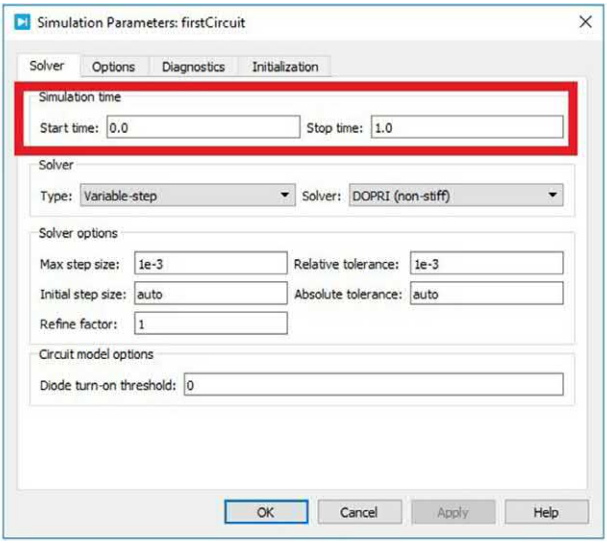

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
V_dc
V: 10
R2
R: 2200
Vm1
V
Scope

FIG. 2.28 Simulation parameter is used to specify the simulation setting (i.e., stop time, solver type, max step size, etc.).

In this example, we want to ask PLECS to simulate the circuit for an interval of length 1 s. To do this, after clicking the "Simulation parameter," we set the "Stop time:" box to 1.0 as shown in Fig. 2.29. Leave the other settings unchanged.

To simulate the schematic, click on "Start" or press the Ctrl+T on your keyboard (Fig. 2.30).


Simulation Parameters: firstCircuit
Solver
Options
Diagnostics
Initialization
Simulation time
Solver
Solver options
Circuit model options
OK
Cancel
Apply
Help

Start time: 0.0
Stop time: 1.0
Type: Variable-step
Solver: DOPRI (non-stiff)
Max step size: 1e-3
Relative tolerance: 1e-3
Initial step size: auto
Absolute tolerance: auto
Refine factor: 1
Diode turn-on threshold: 0

FIG. 2.29 Simulation Parameter window.

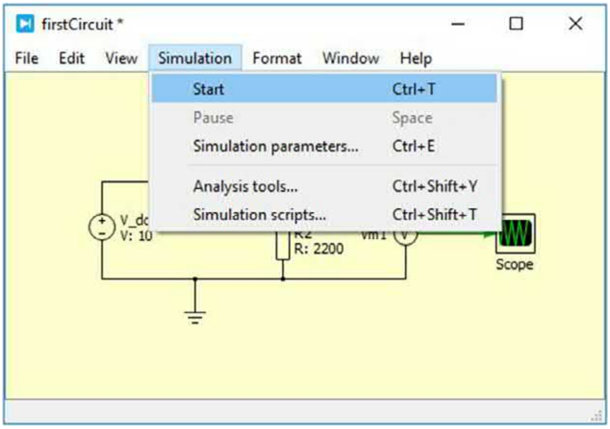

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
V_dc
V: 10
R2
R: 2200
Vm1
V
Scope

FIG. 2.30 Starting the simulation can be done by clicking the Start.

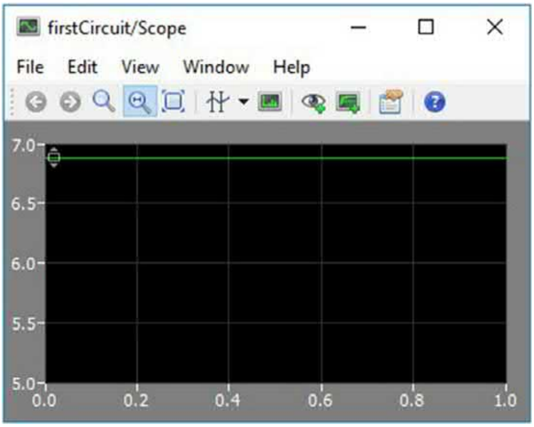

firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.31 Simulation result (R2 voltage).

To see the resistor R2 voltage, double click the scope block. Fig. 2.31 shows the result.

2.1.2 Adding title to the scope

As shown in Fig. 2.31, axes have no titles. In order to add titles to the scope, click on "Scope parameters" icon. After clicking the icon, the window shown in Fig. 2.32 will appears.

Go to the "Plot 1" tab (Fig. 2.33).

Enter the desired title and axis label into the "Title:" and "Axis label:" boxes (Fig. 2.34). After clicking the "OK" button, PLECS adds the entered texts to the graph (Figs. 2.35 and 2.36).

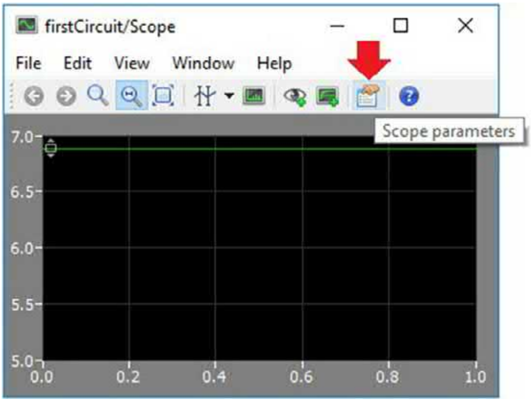
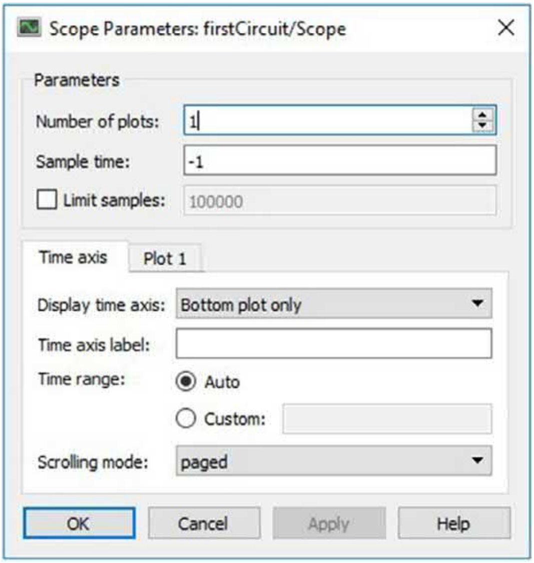

firstCircuit/Scope
File
Edit
View
Window
Help
Scope parameters
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.32 Scope parameters icon.


Scope Parameters: firstCircuit/Scope

Parameters

Number of plots: 1

Sample time: -1

☐ Limit samples: 100000

Time axis | Plot 1

Display time axis: Bottom plot only

Time axis label:

Time range: ● Auto ○ Custom:

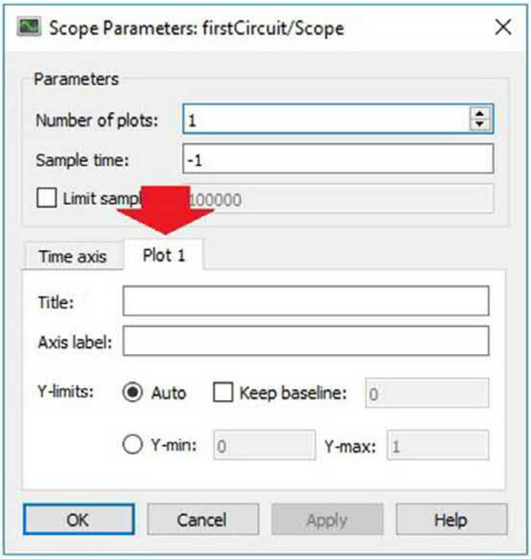
Scrolling mode: paged

OK
Cancel
Apply
Help

FIG. 2.33 Scope parameters window.

2.1.3 Setting the axis limits

You can set the axis lower/upper bound by double clicking on them. For example, if we double click on the Y axis, the windows shown in Fig. 2.37 appears, which lets us to enter the lower and upper value for the Y axis.



Scope Parameters: firstCircuit/Scope

Parameters

Number of plots: 1

Sample time: -1

☐ Limit sampl 100000

Title:

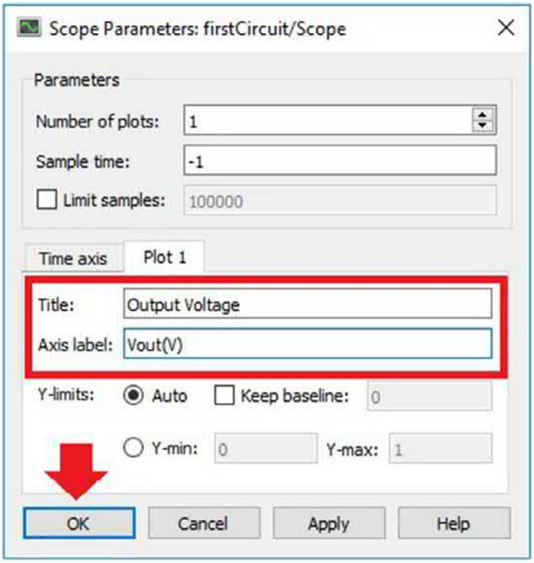
Axis label:

Y-limits: ● Auto ☐ Keep baseline: 0

○ Y-min: 0 Y-max: 1

Time axis
Plot 1
OK
Cancel
Apply
Help

FIG. 2.34 Plot 1 tab of Scope parameters window.



Scope Parameters: firstCircuit/Scope

Parameters

Number of plots: 1

Sample time: -1

☐ Limit samples: 100000

Title: Output Voltage

Axis label: Vout(V)

Y-limits: ● Auto ☐ Keep baseline: 0

○ Y-min: 0 Y-max: 1

Time axis
Plot 1
OK
Cancel
Apply
Help

FIG. 2.35 Specifying the desired title and axis label.

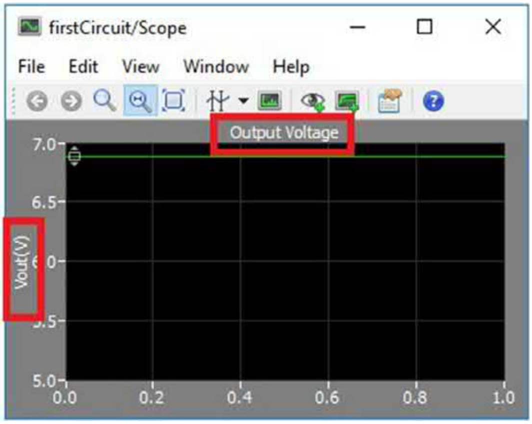
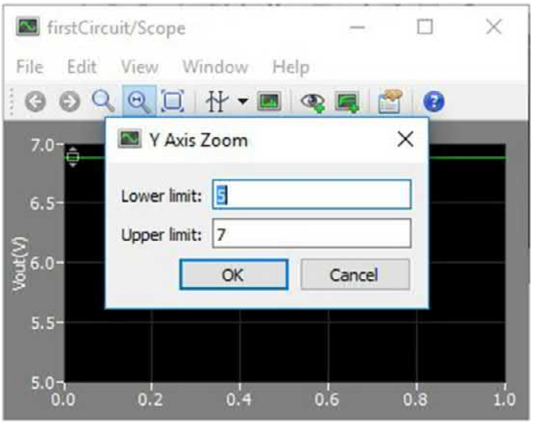

firstCircuit/Scope
File
Edit
View
Window
Help
Output Voltage
Vout(V)
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.36 Specified labels are shown on the scope.


firstCircuit/Scope
File
Edit
View
Window
Help
Y Axis Zoom
OK
Cancel
Vout(V)
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0

Lower limit: 5
Upper limit: 7

FIG. 2.37 Y-Axis Zoom window.

2.1.4 Change the properties of the shown waveform

You can change the properties of the waveform shown in the scope by right clicking on the scope screen and select the "Edit curve properties." (Figs. 2.38 and 2.39)

2.1.5 Reading the values using cursors

You can read the waveform values easily with the cursors (Fig. 2.40).

After clicking the cursors, two cursors are added to the scope. You can bring them to the desired places (Fig. 2.41).

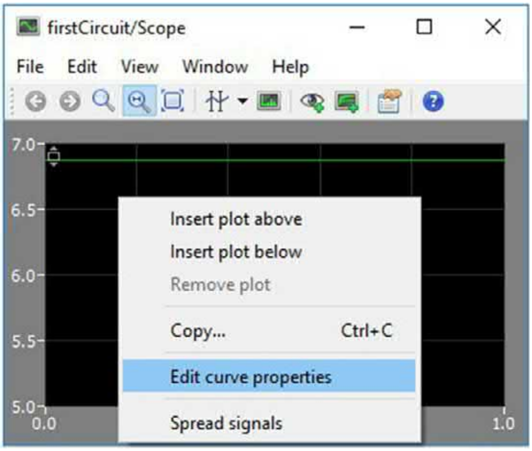
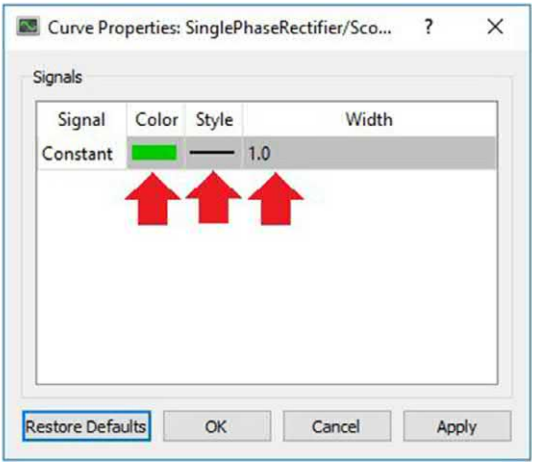

firstCircuit/Scope
File Edit View Window Help
7.0
6.5
6.0
5.5
5.0
0.0
1.0
Insert plot above
Insert plot below
Remove plot
Copy... Ctrl+C
Edit curve properties
Spread signals

FIG. 2.38 "Edit curve properties" is used to specify the desired curve properties.


Curve Properties: SinglePhaseRectifier/Sco...
Signals
Restore Defaults
OK
Cancel
Apply

Signal	Color	Style	Width
Constant	🟩	—	1.0

FIG. 2.39 Specifying the desired curve properties.

Using cursors one can measure the time difference between two cursors, minimum/maximum/average/RMS/THD of waveform captured between two cursors. To do such measurements, you must click the small triangle behind the cursors icon. After clicking the small triangle, a menu will appear (Fig. 2.42).

For example, assume that we want to find the time difference between the two cursors. We must click the "Delta" (see Fig. 2.43) to find the time difference. After clicking the "Delta," PLECS shows the time difference.

Note the lock icon shown in Fig. 2.43. It is an open lock meaning that you can slide each cursor independently (Fig. 2.44).

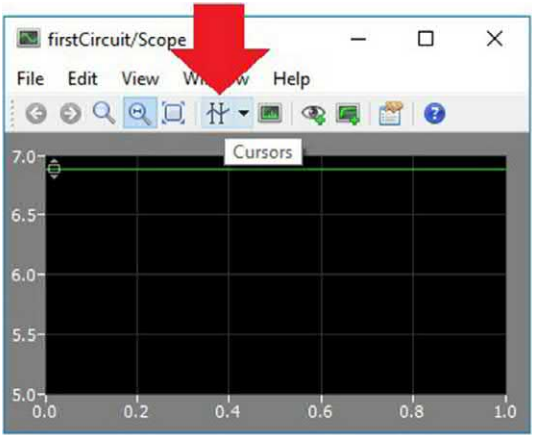
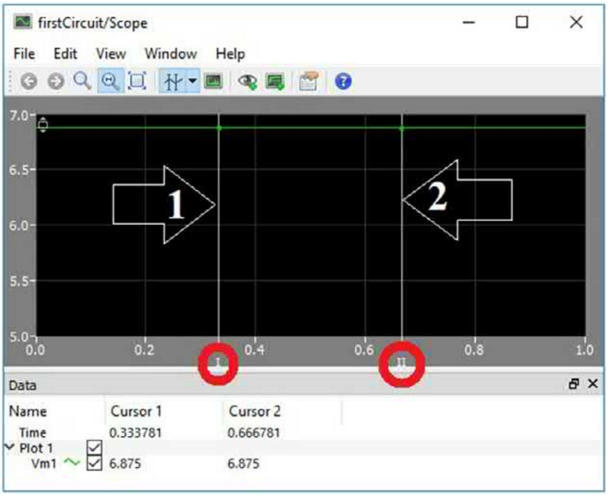

firstCircuit/Scope
File
Edit
View
Wi
w
Help
Cursors
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.40 Cursor icon.


firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0
1
2
I
II
Data

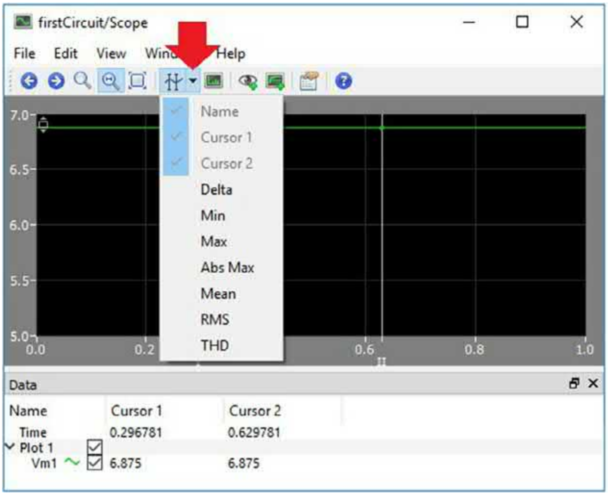
Name		Cursor 1	Cursor 2
Time		0.333781	0.666781
Plot 1	☑		
Vm1	☑	6.875	6.875

FIG. 2.41 After clicking the cursor icon, 2 cursors are appear.

If you click on the lock icon it will change to a close lock. In this case, you can slide either of cursors, but the other one moves as well, i.e., two cursors are dependent. The difference between two cursors is constant and (in this example) is equal to 0.333 s (Fig. 2.45).

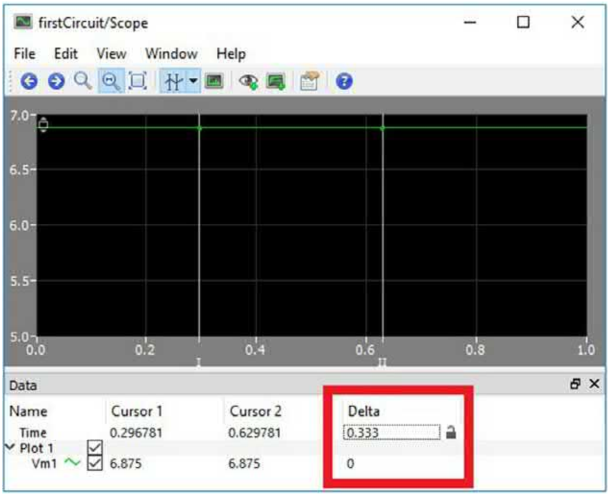
2.1.6 Zoom in/out

Use the magnifier icon to zoom in/out the waveform shown in the scope (Fig. 2.46).


firstCircuit/Scope
File
Edit
View
Win
Help
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
RMS
THD
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.6
0.8
1.0
Data

Name	Cursor 1	Cursor 2
Time	0.296781	0.629781
Plot 1 ☑		
Vm1 ☑	6.875	6.875

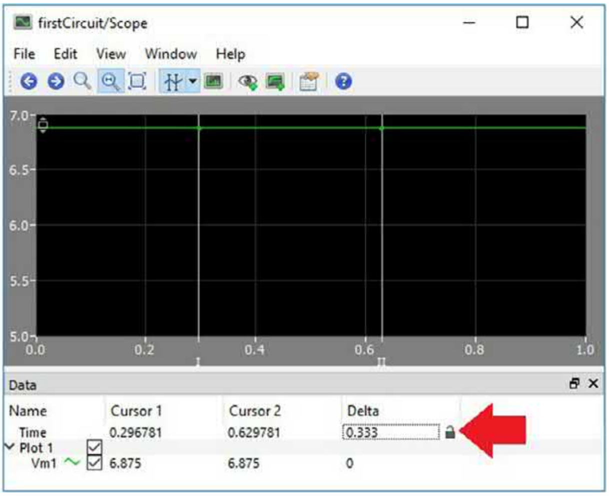
FIG. 2.42 Menu appeared after clicking the small triangle.


firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0
Data

Name	Cursor 1	Cursor 2	Delta
Time	0.296781	0.629781	0.333
Plot 1 ☑			
Vm1 ☑	6.875	6.875	0

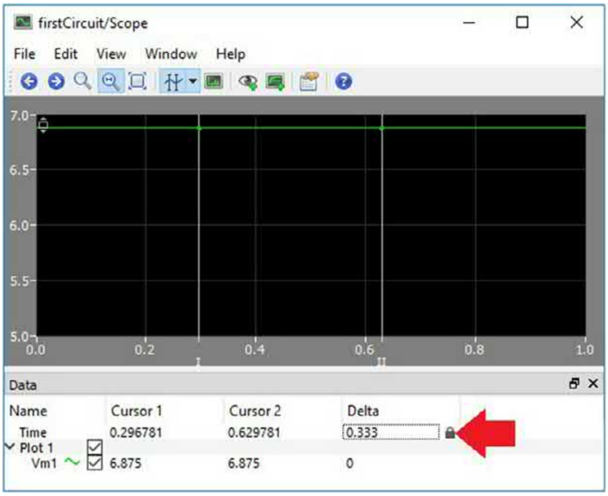
FIG. 2.43 Difference (both time difference and voltage difference) is calculated and shown on the Data window.

For example if you want to see the [0.4 S, 0.6 S] time interval with more detail, use the 🔍 icon. After clicking the 🔍 icon, left click around the 0.4 S and move the mouse pointer toward the 0.6 S. After reaching the desired destination, release the mouse button. After releasing the mouse button, PLECS zoom into the selected region (Figs. 2.47 and 2.48).


firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0
I
II
Data

Name		Cursor 1	Cursor 2	Delta
Time		0.296781	0.629781	0.333
Plot 1	☑			
Vm1	☑	6.875	6.875	0

FIG. 2.44 Open lock.

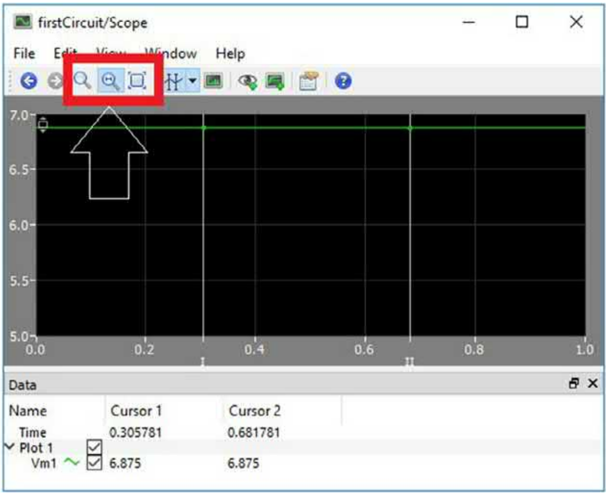

firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0
I
II
Data

Name		Cursor 1	Cursor 2	Delta
Time		0.296781	0.629781	0.333
Plot 1	☑			
Vm1	☑	6.875	6.875	0

FIG. 2.45 Closed lock.

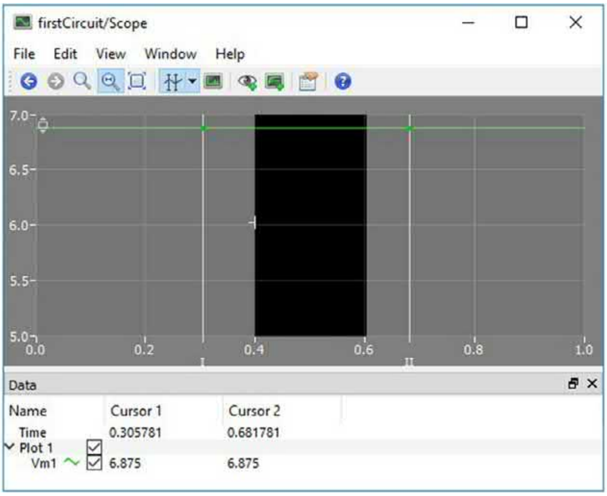
Application of 🔍 icon is not limited to horizontal zooming. You can use it to do vertical zooming as well (Figs. 2.49 and 2.50).

PLECS memorizes the settings of scope each time you do zooming. Use the ← and → icons to move between different views. For example, if you do zooming and you are not happy with the new view, you can return to previous view simply by clicking on the return icon (←) (Fig. 2.51).


firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0
I
II
Data

Name		Cursor 1	Cursor 2
Time		0.305781	0.681781
Plot 1	☑		
Vm1	☑	6.875	6.875

FIG. 2.46 Zoom icons.


firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0
I
II
Data

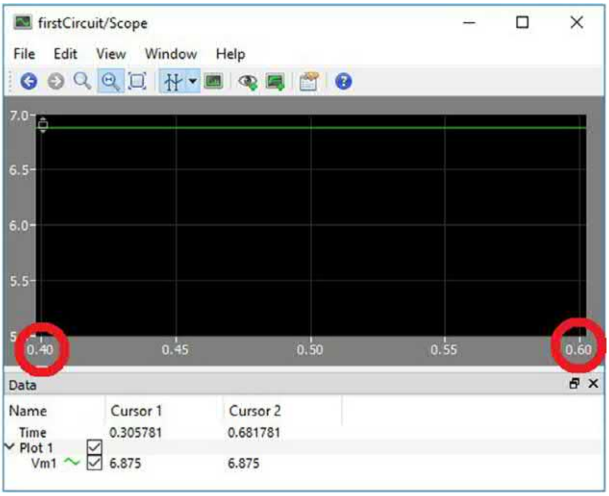
Name		Cursor 1	Cursor 2
Time		0.305781	0.681781
Plot 1	☑		
Vm1	☑	6.875	6.875

FIG. 2.47 Selecting the 0.4–0.6 s interval.

2.1.7 Exporting the scope block waveforms

You can simply export the waveform shown in the scope by clicking "Export" and select the desired format (Fig. 2.52).

If you want to write a report and show the obtained waveforms, use the "as Bitmap…". It makes a .bmp file for you, which you can put it easily into your report. After clicking the "as Bitmap…," the window shown in Fig. 2.53 will


firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5
0.40
0.45
0.50
0.55
0.60
Data

Name		Cursor 1	Cursor 2
Time		0.305781	0.681781
Plot 1	☑		
Vm1	☑	6.875	6.875

FIG. 2.48 PLECS zoom into the selected region.

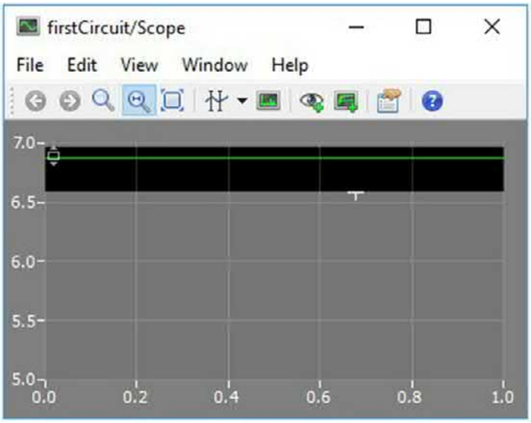

firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.49 Zooming the Y axis.

appear. Enter the desired width and height into the “Width” and “Height” boxes. Use the “Image settings” section to set the produced image quality. Generally, “Resolution (dpi): 72” and “Quality (0-100): 80” is good for most applications, but you can increase them at the cost of produced larger file. For instance, while “Resolution (dpi): 72” and “Quality (0 – 100): 80” produces only 8 KB, “Resolution (dpi): 600” and “Quality (0–100): 80” produces a 152-KB file.

You can copy the waveform shown in the scope into the Windows clipboard. This capability is useful when you want to edit the obtained waveform in a graphic software. In order to copy the waveform into the clipboard, click The “Copy…” (Fig. 2.54).

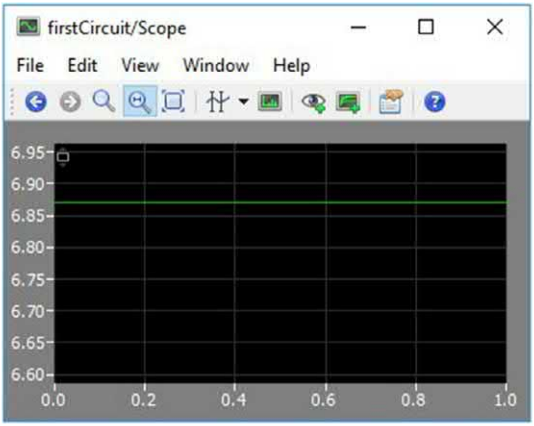
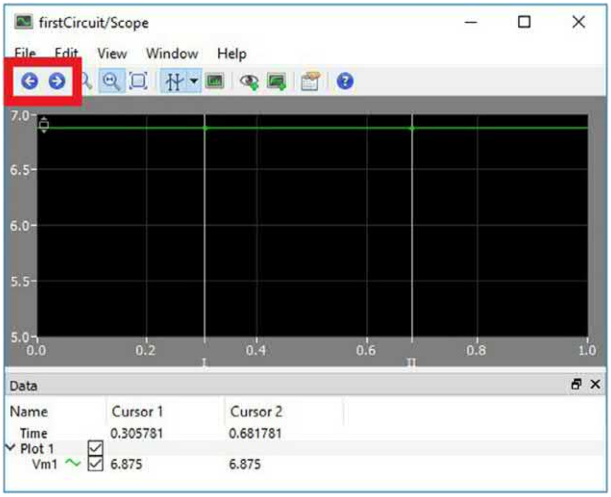

firstCircuit/Scope
File
Edit
View
Window
Help
6.95
6.90
6.85
6.80
6.75
6.70
6.65
6.60
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.50 Zoomed Y axis.


firstCircuit/Scope
File
Edit
View
Window
Help
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0
I
II
Data

Name		Cursor 1	Cursor 2
Time		0.305781	0.681781
Plot 1	☑		
Vm1	☑	6.875	6.875

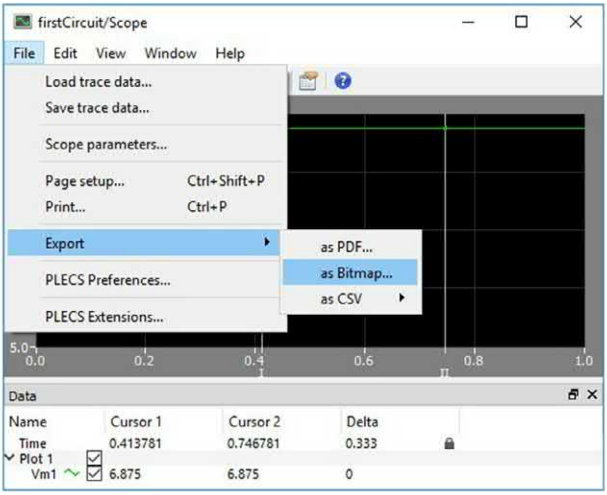
FIG. 2.51 Using the arrows, you can move between the different views.

After clicking the "Edit…," the window shown in Fig. 2.55 will appear. Do the desired settings and click on the "Copy to Clipboard" button.

2.1.8 Exporting the drawn schematic

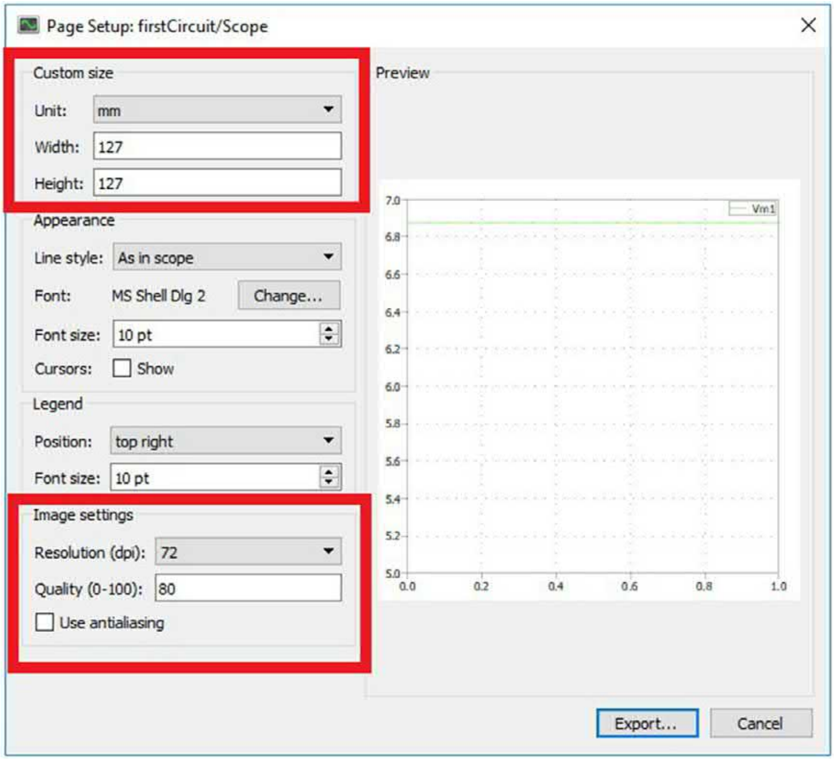
You can export the drawn schematic easily with the "Export schematic…" (Fig. 2.56).

After clicking the "Export schematic…," the window shown in Fig. 2.57 will appear. Select the desired resolution and press the "OK" button. After


firstCircuit/Scope
File
Edit
View
Window
Help
Load trace data...
Save trace data...
Scope parameters...
Page setup... Ctrl+Shift+P
Print... Ctrl+P
Export
PLECS Preferences...
PLECS Extensions...
as PDF...
as Bitmap...
as CSV
5.0
0.0
0.2
0.4
0.6
0.8
1.0
I
II
Data

Name		Cursor 1	Cursor 2	Delta
Time		0.413781	0.746781	0.333
Plot 1	☑			
Vm1	☑	6.875	6.875	0

FIG. 2.52 Exporting the scope graphs as Bitmap(.bmp) file.


Page Setup: firstCircuit/Scope
Preview
Change...
7.0
6.8
6.6
6.4
6.2
6.0
5.8
5.6
5.4
5.2
5.0
0.0
0.2
0.4
0.6
0.8
1.0
Vm1
Export...
Cancel

Custom size
Unit: mm
Width: 127
Height: 127

Appearance
Line style: As in scope
Font: MS Shell Dlg 2
Font size: 10 pt
Cursors: ☐ Show

Legend
Position: top right
Font size: 10 pt

Image settings
Resolution (dpi): 72
Quality (0-100): 80
☐ Use antialiasing

FIG. 2.53 Page Setup window.

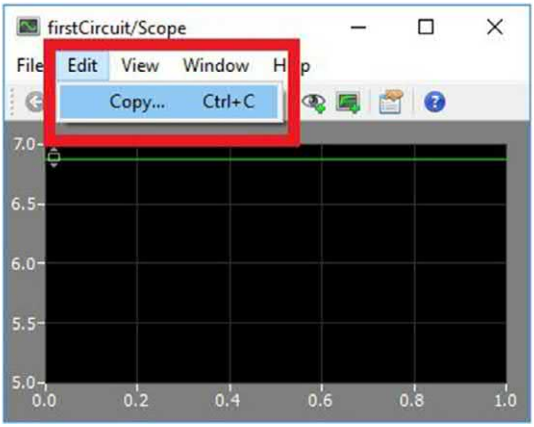
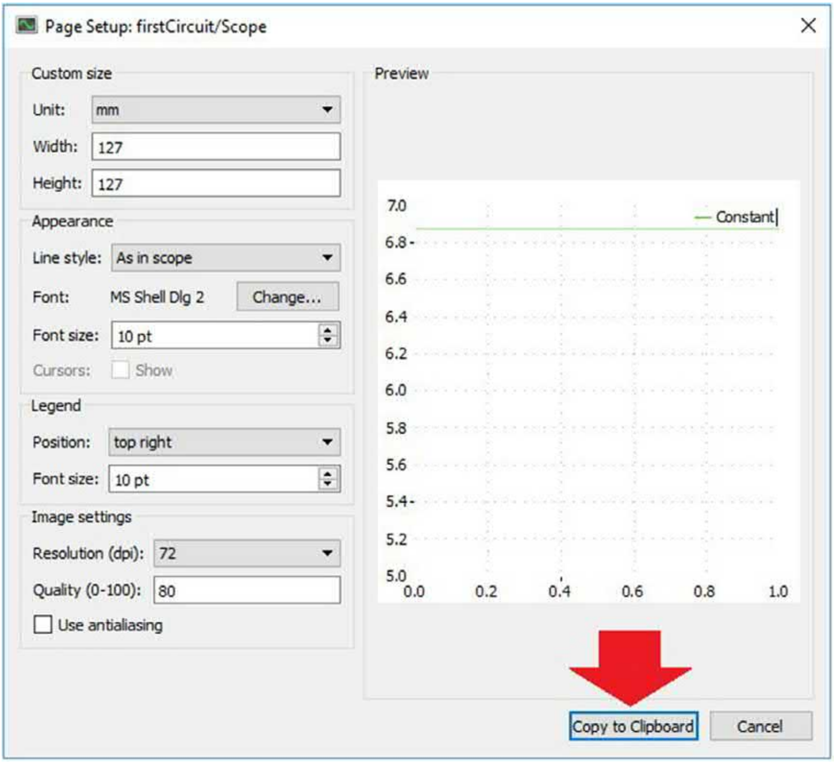

firstCircuit/Scope
File
Edit
View
Window
Help
Copy...
Ctrl+C
7.0
6.5
6.0
5.5
5.0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.54 Copying the scope graphs into the Windows clipboard.



Page Setup: firstCircuit/Scope

Custom size

Unit: mm

Width: 127

Height: 127

Appearance

Line style: As in scope

Font: MS Shell Dlg 2

Font size: 10 pt

Cursors: ☐ Show

Legend

Position: top right

Font size: 10 pt

Image settings

Resolution (dpi): 72

Quality (0-100): 80

☐ Use antialiasing

Change...
Preview
7.0
6.8
6.6
6.4
6.2
6.0
5.8
5.6
5.4
5.2
5.0
0.0
0.2
0.4
0.6
0.8
1.0
Constant
Copy to Clipboard
Cancel

FIG. 2.55 Page Setup window.

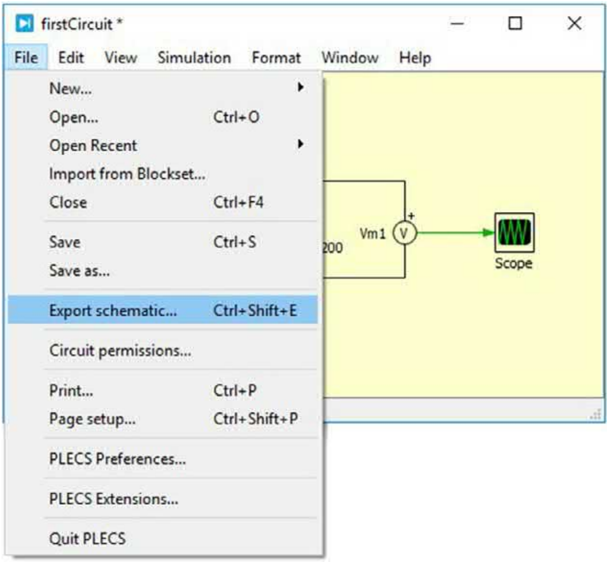
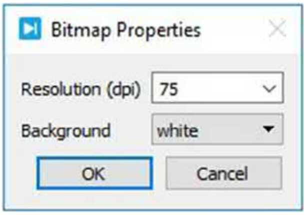

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
New...
Open... Ctrl+O
Open Recent
Import from Blockset...
Close Ctrl+F4
Save Ctrl+S
Save as...
Export schematic... Ctrl+Shift+E
Circuit permissions...
Print... Ctrl+P
Page setup... Ctrl+Shift+P
PLECS Preferences...
PLECS Extensions...
Quit PLECS
200
Vm1
V
+
Scope

FIG. 2.56 Exporting the schematic as graphic file.


Bitmap Properties
OK
Cancel

Resolution (dpi) 75

Background white

FIG. 2.57 Bitmap Properties window.

pressing the "OK" button, you must specify a place to save the file. Exported schematic is shown in Fig. 2.58. You can use the exported graphic file for the purpose of documentation.

You can even copy the schematic to Windows clipboard. To do this, use the "Copy as image." You can paste the image in the graphical or word processor software (Fig. 2.59).

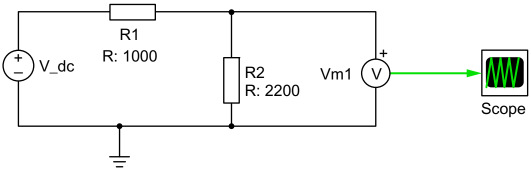

+
−
V_dc
R1
R: 1000
R2
R: 2200
Vm1
V
+
Scope

FIG. 2.58 Exported graphic file.

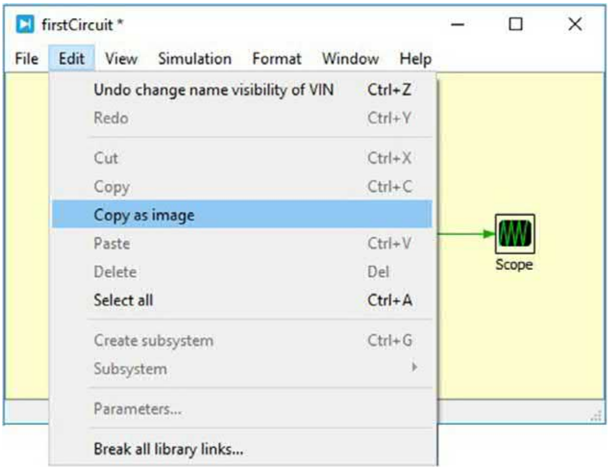

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
Undo change name visibility of VIN Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Copy as image
Paste Ctrl+V
Delete Del
Select all Ctrl+A
Create subsystem Ctrl+G
Subsystem
Parameters...
Break all library links...
Scope

FIG. 2.59 Copying the schematic into the Windows clipboard.

2.1.9 Display block

DC quantities can be displayed easily with the aid of "Display" block (Figs. 2.60 and 2.61).

2.1.10 Changing the block names

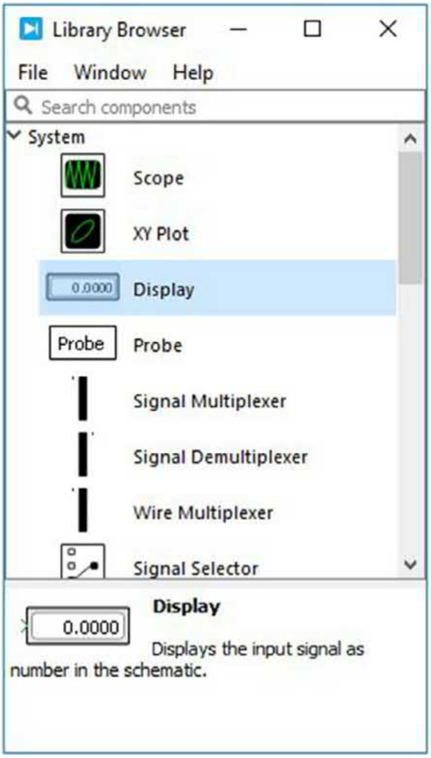
Selection of meaningful names for components is an important issue since it simplifies the understanding of circuit. You can change the default components names easily by double clicking on their name and writing the new name (Figs. 2.62 and 2.63).

2.1.11 Hiding the block names

You can ask PLECS not to show the component name if you prefer to do so. To hide the component name, simply right click on the component and uncheck the "Show name." After unchecking the "Show name," the components name will be hidden (Figs. 2.64 and 2.65).

2.1.12 Adding text to the schematic

You can add text to the schematic by double clicking on the schematic and writing the desired text into the appeared box (Fig. 2.66).


Library Browser
File Window Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
0.0000

Display

Displays the input signal as number in the schematic.

FIG. 2.60 Display block.

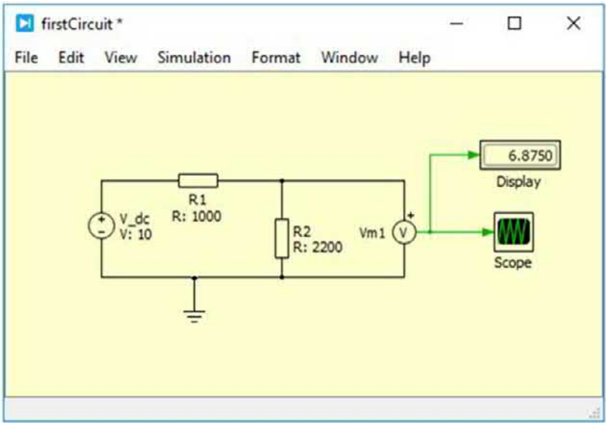

firstCircuit *
File Edit View Simulation Format Window Help
R1
R: 1000
V_dc
V: 10
R2
R: 2200
Vm1
V
6.8750
Display
Scope

FIG. 2.61 Adding the display block to the schematic.

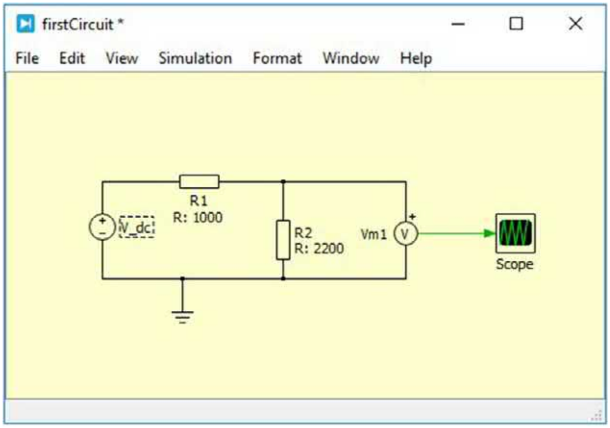

firstCircuit *
File Edit View Simulation Format Window Help
R1
R: 1000
V_dc
R2
R: 2200
Vm1
V
Scope

FIG. 2.62 Renaming the components.

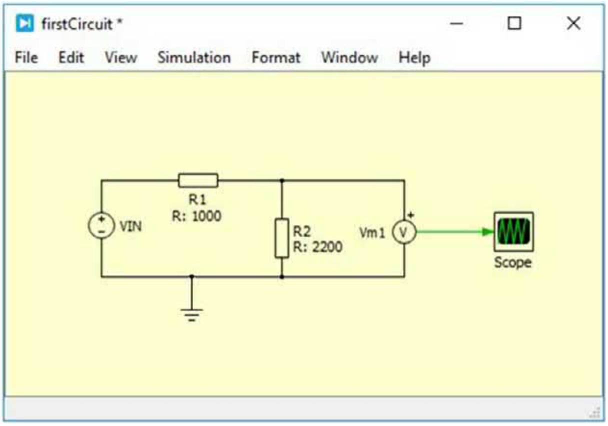

firstCircuit *
File Edit View Simulation Format Window Help
R1
R: 1000
VIN
R2
R: 2200
Vm1
V
Scope

FIG. 2.63 Schematic with new names.

2.1.13 Ammeter block

You can add an ammeter to the circuit in order to see the circuit current. To do so, first click on the wire connecting the source to the resistor R1. Press the keyboard "Delete" key to remove that wire (Figs. 2.67 and 2.68).

After wire has been removed, add an ammeter in its place. Connect the output port of ammeter to a scope in order to see the waveform (Figs. 2.69–2.71).

It is a good practice to select meaningful names for scopes. This makes the understanding of circuit easy. You can change the scope labels easily by double clicking on the current label ("Scope" and "Scope1" in Fig. 2.72) and writing the new label in the opened box.

If we run the simulation we obtain the result shown in Fig. 2.73.

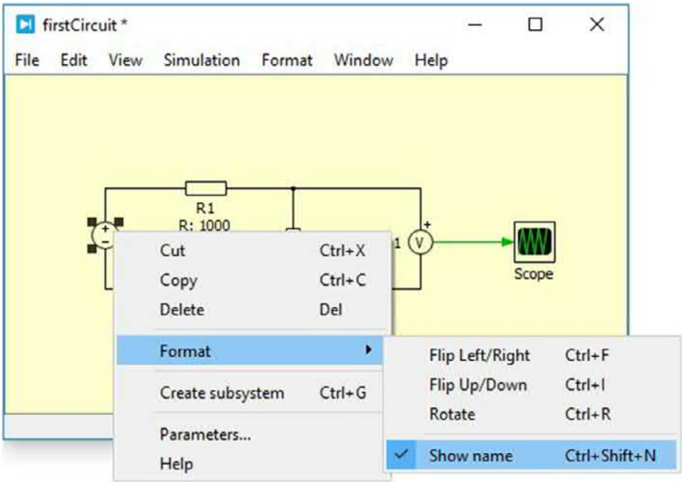

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 1000
Scope
Cut Ctrl+X
Copy Ctrl+C
Delete Del
Format
Create subsystem Ctrl+G
Parameters...
Help
Flip Left/Right Ctrl+F
Flip Up/Down Ctrl+I
Rotate Ctrl+R
Show name Ctrl+Shift+N

FIG. 2.64 Show/hide the block names.

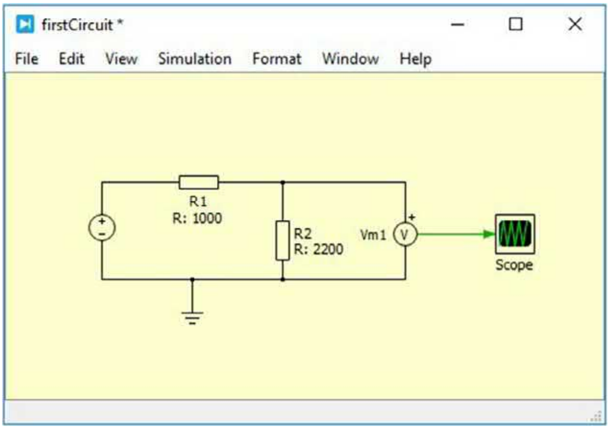

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 1000
R2
R: 2200
Vm1
Scope

FIG. 2.65 Input source name is hided.

If we use the icon, we can obtain a closer look of the current waveform. As expected, result is 3.12 mA (Fig. 2.74).

2.1.14 Wire colors

Power electronics converters are composed of a power circuit and a control algorithm. PLECS shows the power circuit components in black and control (or measurement) signals in green (Fig. 2.75).

You cannot connect components designed for power circuit to control (or measurement) signals (Fig. 2.76).

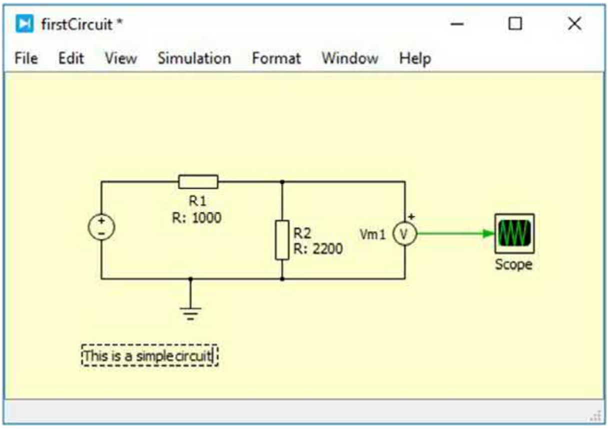

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 1000
R2
R: 2200
Vm1
V
Scope
This is a simple circuit

FIG. 2.66 Adding text to the schematic.

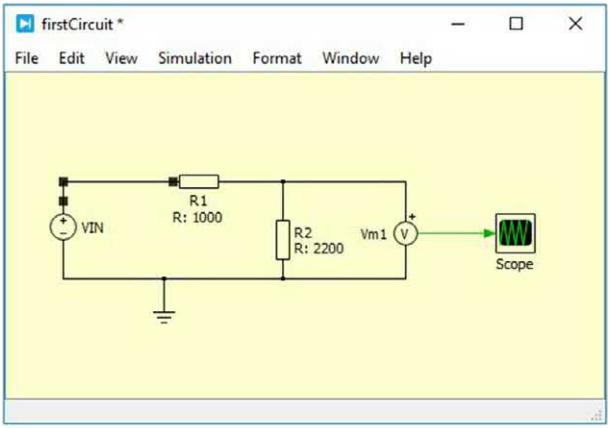

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
VIN
R1
R: 1000
R2
R: 2200
Vm1
V
Scope

FIG. 2.67 You can select a wire by clicking on it.

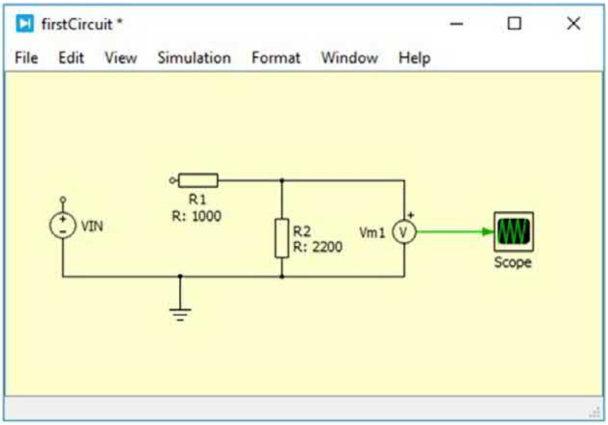
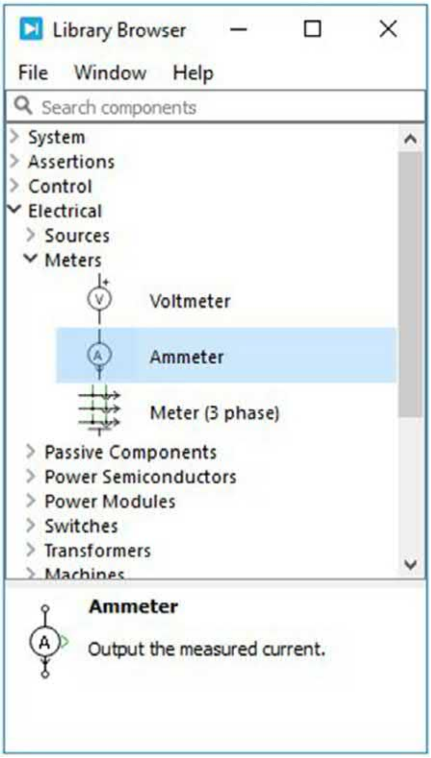

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
VIN
R1
R: 1000
R2
R: 2200
Vm1
V
Scope

FIG. 2.68 Remove the selected wire by pressing the Delete key on keyboard.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Voltmeter
Ammeter
Meter (3 phase)
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines
Ammeter

Output the measured current.

FIG. 2.69 Ammeter block.

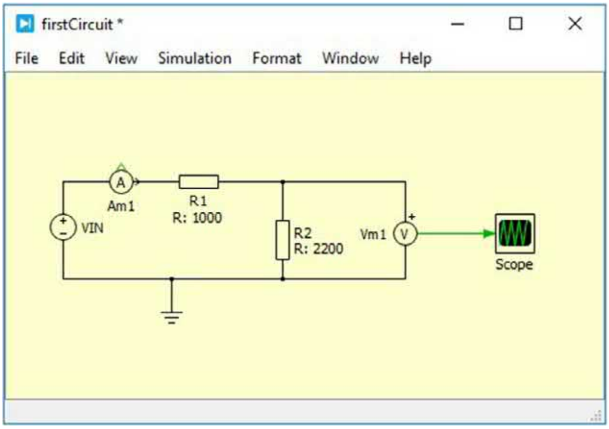

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
Am1
VIN
R1
R: 1000
R2
R: 2200
Vm1
Scope

FIG. 2.70 Adding an ammeter block to the schematic.

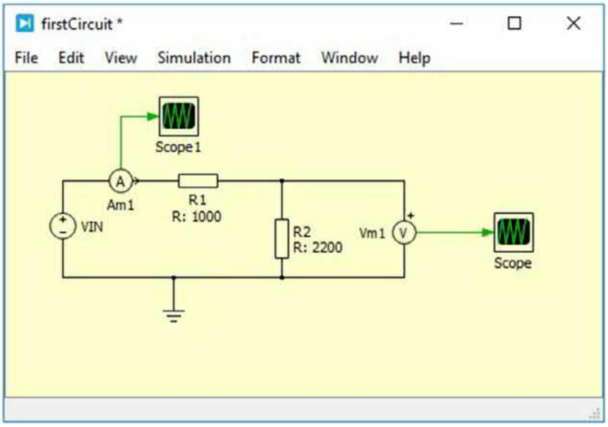

firstCircuit *
File Edit View Simulation Format Window Help
Scope 1
A
Am1
R1
R: 1000
VIN
R2
R: 2200
Vm1
V
Scope

FIG. 2.71 Connecting the ammeter block to the Scope 1 block.

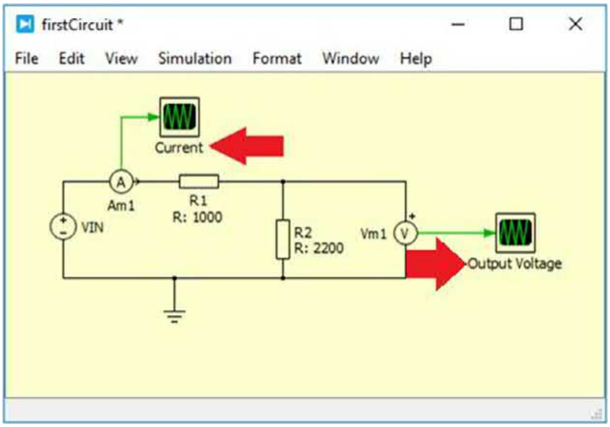

firstCircuit *
File Edit View Simulation Format Window Help
Current
A
Am1
R1
R: 1000
VIN
R2
R: 2200
Vm1
V
Output Voltage

FIG. 2.72 Renaming the scopes.

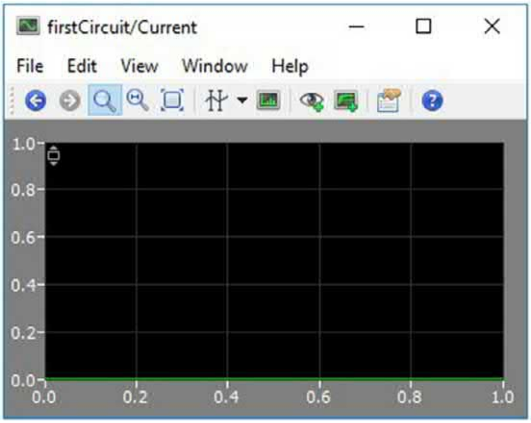

firstCircuit/Current
File Edit View Window Help
1.0
0.8
0.6
0.4
0.2
0.0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.73 Current scope waveform.

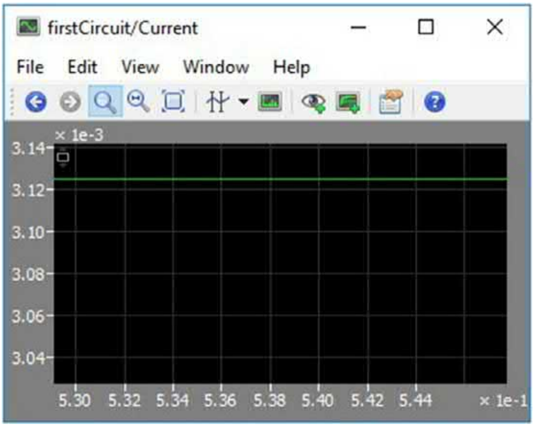

firstCircuit/Current
File
Edit
View
Window
Help
× 1e-3
3.14
3.12
3.10
3.08
3.06
3.04
5.30
5.32
5.34
5.36
5.38
5.40
5.42
5.44
× 1e-1

FIG. 2.74 Zooming the Current scope waveform. Circuit current is about 3.12 mA.

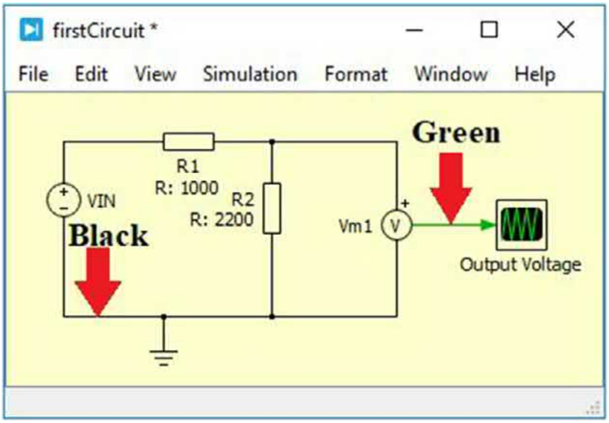

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
Green
R1
R: 1000
R2
R: 2200
VIN
Black
Vm1
Output Voltage

FIG. 2.75 PLECS use different colors to show wires.

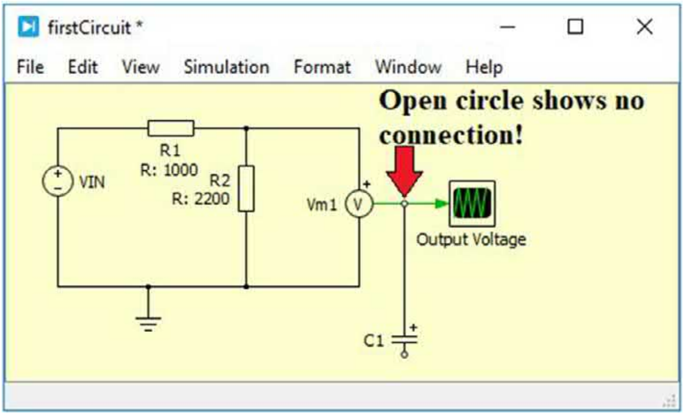
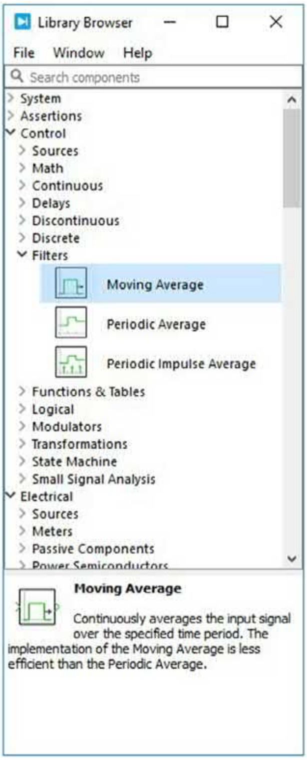

firstCircuit *
File
Edit
View
Simulation
Format
Window
Help
Open circle shows no connection!
R1
R: 1000
R2
R: 2200
VIN
Vm1
Output Voltage
C1

FIG. 2.76 You cannot connect power circuit components to control signals.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Moving Average
Periodic Average
Periodic Impulse Average
Functions & Tables
Logical
Modulators
Transformations
State Machine
Small Signal Analysis
Electrical
Sources
Meters
Passive Components
Power Semiconductors

Moving Average

Continuously averages the input signal over the specified time period. The implementation of the Moving Average is less efficient than the Periodic Average.

FIG. 2.77 Moving Average block.

Green signals (control or measurement signals) can connect to blocks, which can process them. For example, you can connect output of a voltmeter to a "Moving Average" block since it can process the measured voltage but you cannot connect the output of a voltmeter to a capacitor (Figs. 2.77 and 2.78).

2.2 Example 2.2: RC circuit analysis

We want to simulate a simple RC circuit as our second example.

2.2.1 Preparing the simulation

Assume the schematic shown in Fig. 2.79. Capacitor can be found in Electrical > Passive Components as shown in Fig. 2.80. Used components settings are shown in Figs. 2.81–2.83.

We want to simulate the circuit for an interval of length 10 ms (Fig. 2.84). To do this, we click the "Simulation parameters…" and in the appeared window,

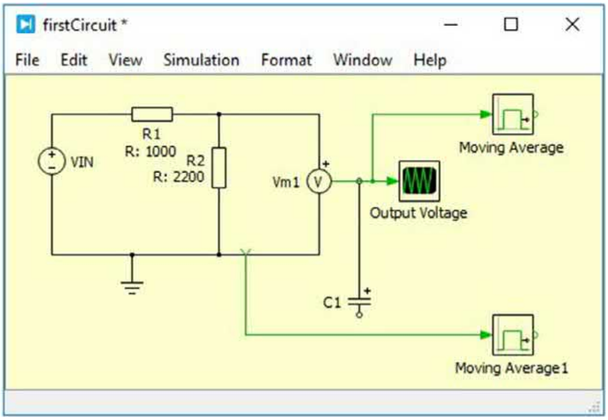

firstCircuit *
File Edit View Simulation Format Window Help
R1
R: 1000
VIN
R2
R: 2200
Vm1
Moving Average
Output Voltage
C1
Moving Average1

FIG. 2.78 Moving average input expect control signal, so you can connect it to the output of voltmeter block. If the input port of a block is drawn in green, you can connect it only to green (control signal) wires.

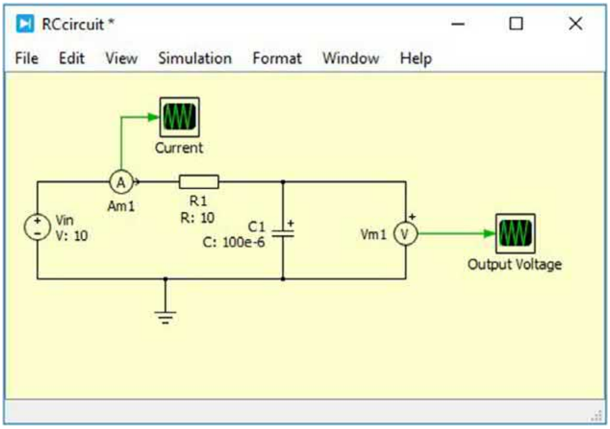

RCcircuit *
File Edit View Simulation Format Window Help
Current
Am1
R1
R: 10
Vin
V: 10
C1
C: 100e-6
Vm1
Output Voltage

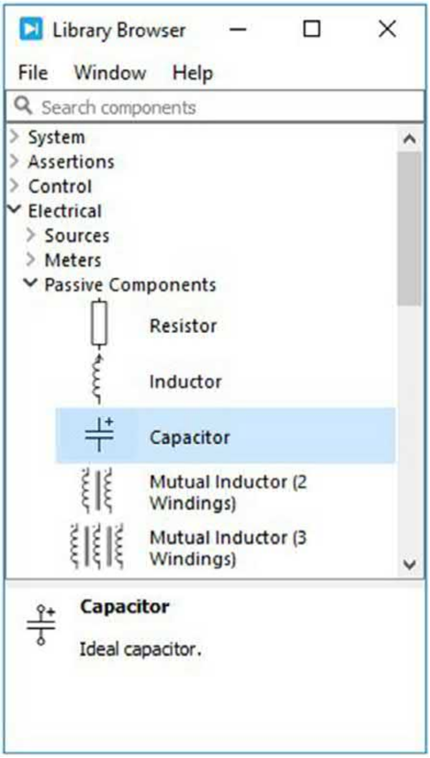
FIG. 2.79 Schematic of the RC circuit.

set the "Stop time:" text box to 10e-3 as shown in Fig. 2.85. Do the other setting like that shown in Fig. 2.85.

After running the simulation, the waveform shown in Figs. 2.86 and 2.87 will be obtained. As you see, the waveforms are not very smooth, they seem to be piecewise linear.

Close the scopes and click the "Simulation parameters…" again (see Fig. 2.84). Reduce the "Max step size:" box to 1e-6, which means 10^{-6} (Fig. 2.88).

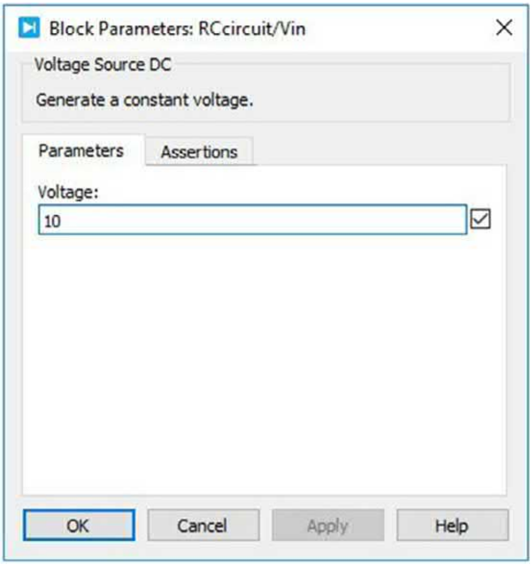
If we run the simulation, we obtain the waveforms shown in Figs. 2.89 and 2.90. This time the waveforms are more smooth (But simulation takes more time to complete).


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Resistor
Inductor
Capacitor
Mutual Inductor (2 Windings)
Mutual Inductor (3 Windings)

Capacitor

Ideal capacitor.

FIG. 2.80 Capacitor block.


Parameters
Assertions
OK
Cancel
Apply
Help

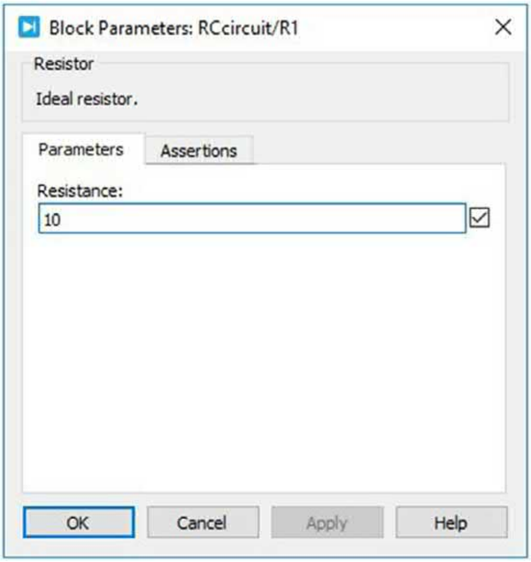
Block Parameters: RCcircuit/Vin

Voltage Source DC

Generate a constant voltage.

Voltage: 10 ☑

FIG. 2.81 DC voltage source settings.


Block Parameters: RCcircuit/R1

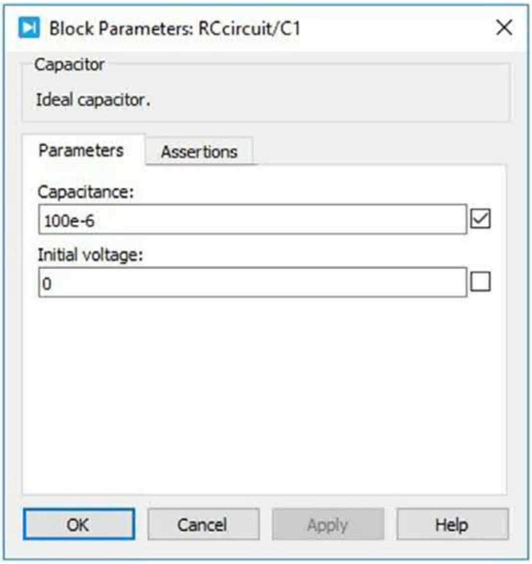
Resistor

Ideal resistor.

Resistance: 10 ☑

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 2.82 Resistor settings.


Block Parameters: RCcircuit/C1

Capacitor

Ideal capacitor.

Capacitance: 100e-6 ☑

Initial voltage: 0 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 2.83 Capacitor settings.

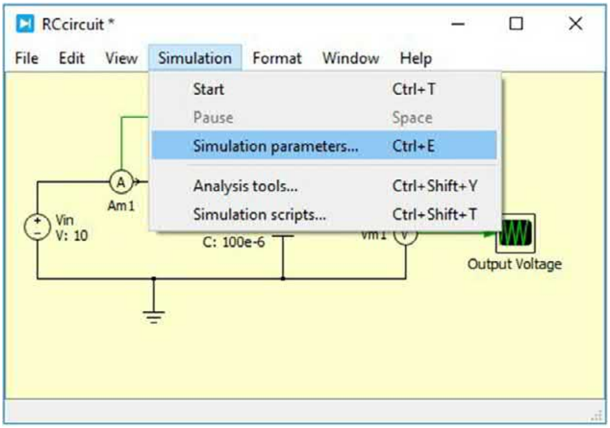
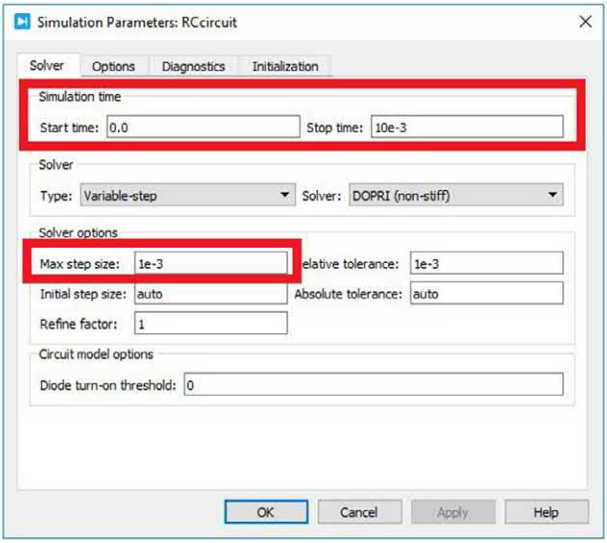

RCcircuit *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
A
Am1
Vin
V: 10
C: 100e-6
Vm1
V
Output Voltage

FIG. 2.84 Simulation parameters.


Simulation Parameters: RCcircuit
Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

Simulation time

Start time: 0.0

Stop time: 10e-3

Solver

Type: Variable-step

Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-3

elative tolerance: 1e-3

Initial step size: auto

Absolute tolerance: auto

Refine factor: 1

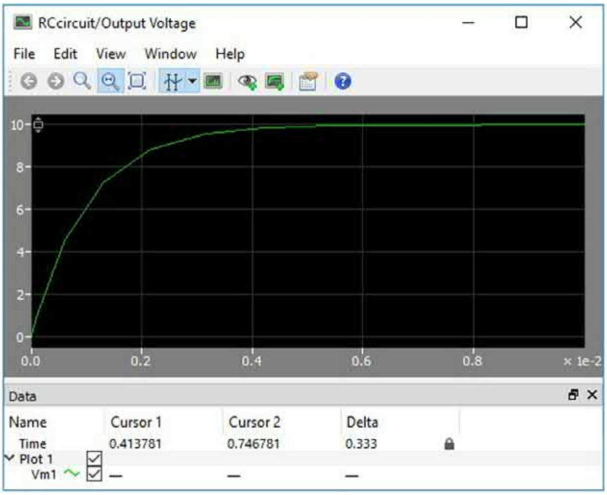
Circuit model options

Diode turn-on threshold: 0

FIG. 2.85 Simulation settings.

Circuit differential equation (which is a continuous-time equation) must be discretized to be solvable by a digital computer. When you discretize the equation with smaller values (smaller value in “Max step size:” box in Fig. 2.88), you increase the accuracy of response but you increase the required computations as well. You can decrease slightly the “Max step size:” value when you obtain piecewise-like waveforms.

There is no need to change the “Solver” section (see Fig. 2.88) for normal applications. See [1] for more information about the solver.


RCcircuit/Output Voltage
File
Edit
View
Window
Help
10
8
6
4
2
0
0.0
0.2
0.4
0.6
0.8
× 1e-2
Data

Name		Cursor 1	Cursor 2	Delta
Time		0.413781	0.746781	0.333
Plot 1	☑			
Vm1	☑	—	—	—

FIG. 2.86 Capacitor voltage waveform.

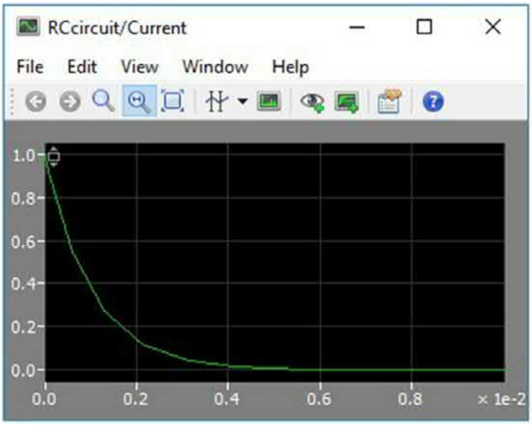
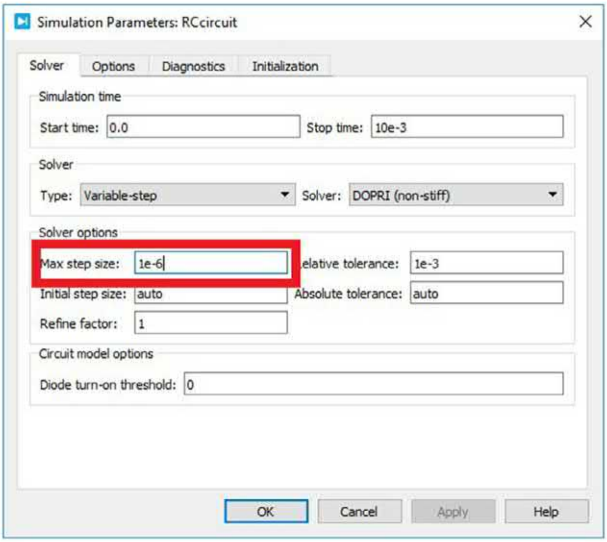

RCcircuit/Current
File
Edit
View
Window
Help
1.0
0.8
0.6
0.4
0.2
0.0
0.0
0.2
0.4
0.6
0.8
× 1e-2

FIG. 2.87 Circuit current waveform.

2.2.2 Specifying the initial condition

You can simulate the circuit with the desired initial conditions. Just double click the capacitor and set the "Initial voltage" box (Figs. 2.91 and 2.92). Here we assume the capacitor initial voltage is 2 V. The capacitor has a terminal shown with a + sign. When you enter "2" in the "Initial voltage:" box, it means $V_{the\ terminal\ with+sign} - V_{the\ terminal\ without+sign} = 2\ V$.


Simulation Parameters: RCcircuit

Simulation time

Start time: 0.0
Stop time: 10e-3

Solver

Type: Variable-step
Solver: DOPRI (non-stiff)

Solver options

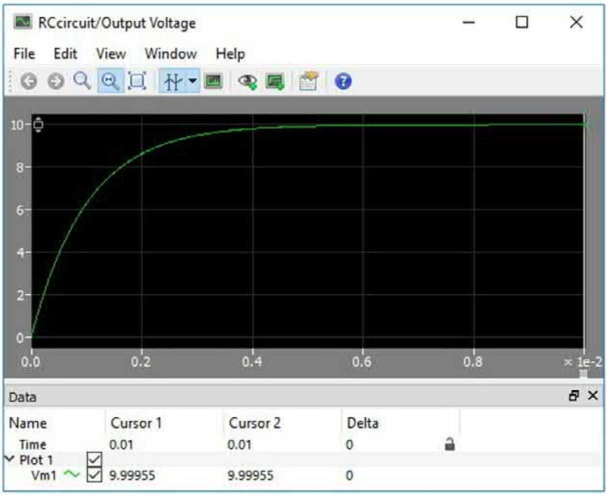
Max step size: 1e-6
Relative tolerance: 1e-3
Initial step size: auto
Absolute tolerance: auto
Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

FIG. 2.88 Max step size text box.


RCcircuit/Output Voltage
File
Edit
View
Window
Help
10
8
6
4
2
0
0.0
0.2
0.4
0.6
0.8
× 1e-2
Data

Name	Cursor 1	Cursor 2	Delta
Time	0.01	0.01	0
Plot 1 ☑			
Vm1 ☑	9.99955	9.99955	0

FIG. 2.89 Capacitor voltage.

If we run the simulation, we obtain the result shown in Fig. 2.93 for capacitor voltage. Note that the voltage starts to increase from 2 V.

You can set the initial current of inductors as well, if your circuit contains any inductors. Just double click on the inductor you want and set its “Initial current:” box. Inductors contain a small arrow, which shows the positive direction.

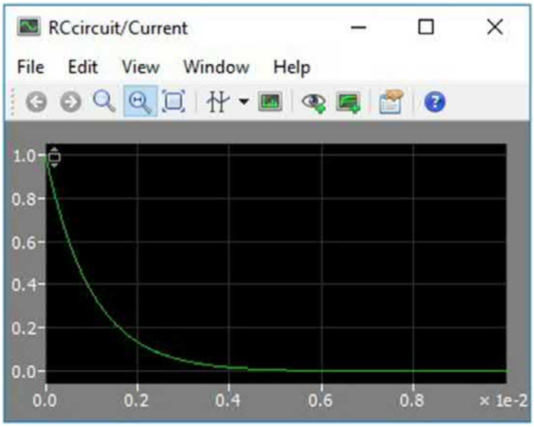
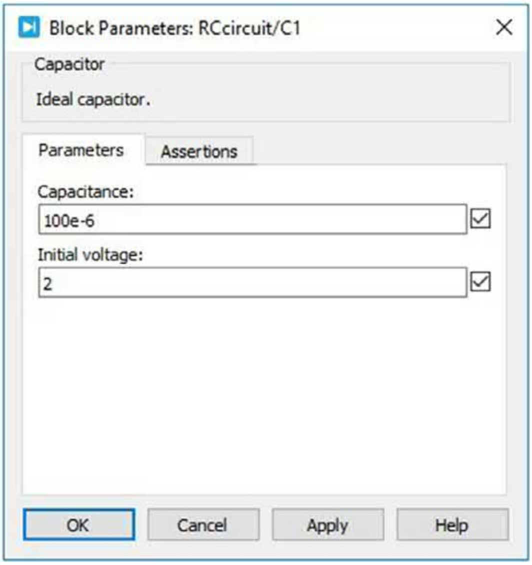


FIG. 2.90 Circuit current.


Block Parameters: RCcircuit/C1

Capacitor

Ideal capacitor.

Capacitance: 100e-6 ☑

Initial voltage: 2 ☑

FIG. 2.91 Setting the capacitor initial voltage to 2 V.

Initial current considers positive when it is in the direction of small arrow (Fig. 2.94).

2.2.3 Showing two or more waveforms simultaneously on the same axis

In PLECS, you can see two or more waveforms on the same axis. This makes comparison possible. The process of showing different waveforms on the same

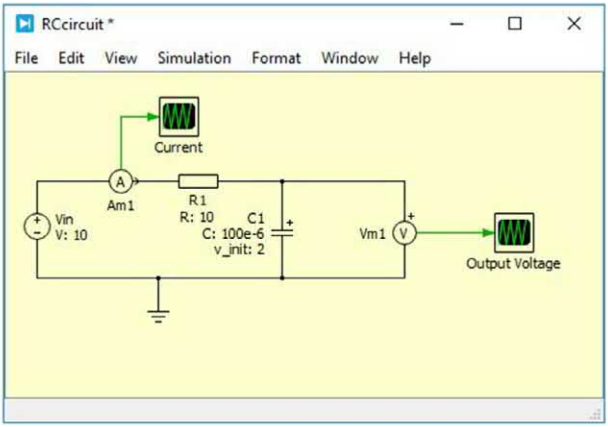
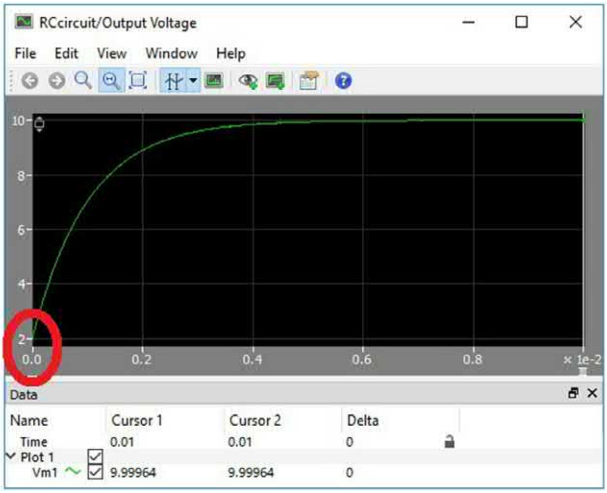

RCcircuit *
File
Edit
View
Simulation
Format
Window
Help
Current
Am1
R1
R: 10
C1
C: 100e-6
v_init: 2
Vin
V: 10
Vm1
Output Voltage

FIG. 2.92 Schematic after setting the capacitor initial voltage.


RCcircuit/Output Voltage
File
Edit
View
Window
Help
10
8
6
4
2
0.0
0.2
0.4
0.6
0.8
× 1e-2
Data

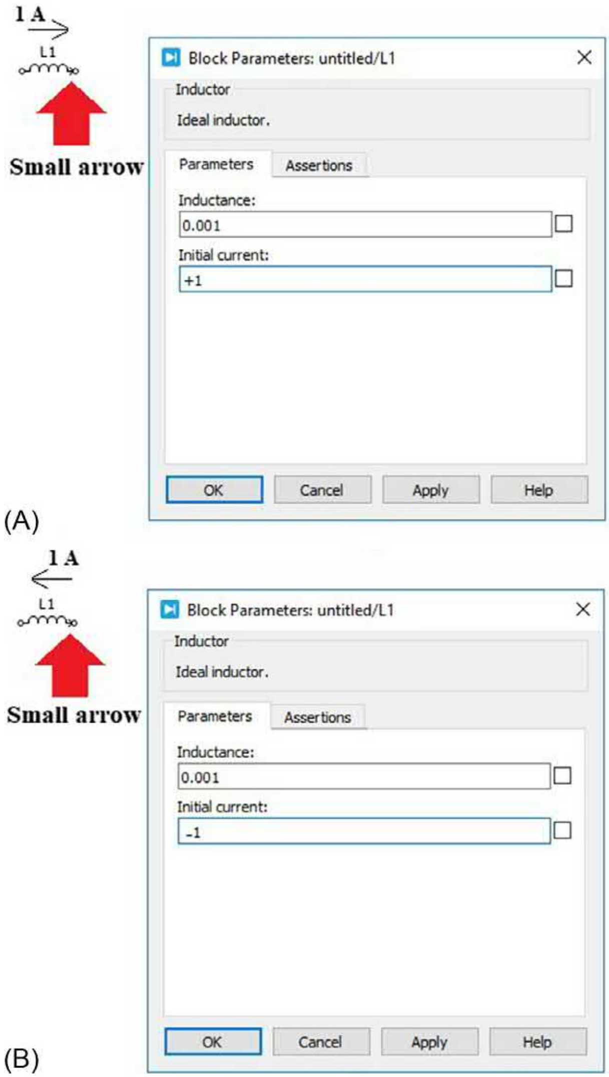
Name	Cursor 1	Cursor 2	Delta
Time	0.01	0.01	0
Plot 1 ☑			
Vm1 ☑	9.99964	9.99964	0

FIG. 2.93 Capacitor voltage waveform.

axis is shown with the aid of an example. Consider a schematic like that shown in Fig. 2.95. The RC circuit is stimulated with a sinusoidal voltage source. We want to see both the input voltage source and capacitor voltage on the same axis. Input sinusoidal voltage source place and settings are shown in Figs. 2.96 and 2.97, respectively.

Add two voltmeters to the circuit as shown in Fig. 2.98.

Instead of using two scopes to show the outputs of two voltmeter block, we use a single scope, which has a multiplexer in its input. Multiplexer block can be found in the "System" section of "Library Browser." (Figs. 2.99 and 2.100)


1 A
L1
Small arrow
(A)

Block Parameters: untitled/L1

Inductor

Ideal inductor.

Parameters
Assertions

Inductance: 0.001 ☐

Initial current: +1 ☐

OK
Cancel
Apply
Help

1 A
L1
Small arrow
(B)

Block Parameters: untitled/L1

Inductor

Ideal inductor.

Parameters
Assertions

Inductance: 0.001 ☐

Initial current: -1 ☐

OK
Cancel
Apply
Help

FIG. 2.94 Setting the inductor initial current. (A) Initial current and the small arrow on the inductor symbol have the same direction (B) Initial current and the small arrow on the inductor symbol have opposite direction.

Double click on the multiplexer block placed in the schematic and decrease its inputs to 2 (Fig. 2.101).

Connect the voltmeter blocks outputs to the input of multiplexer block (Fig. 2.102).

Connect the output of the multiplexer block to a scope (Fig. 2.103).

If we run the simulation, we obtain the result shown in Fig. 2.104. As shown the two waveforms are shown on the same axis.

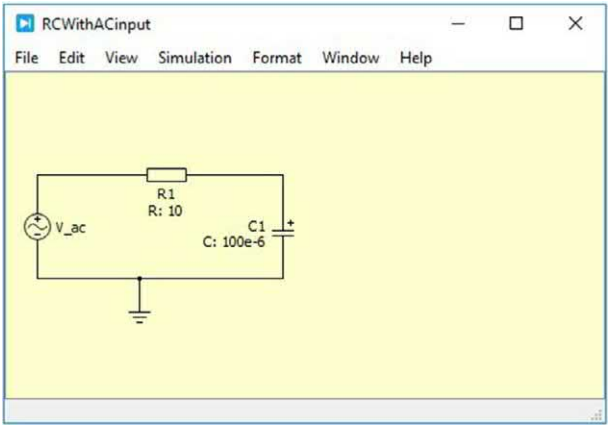
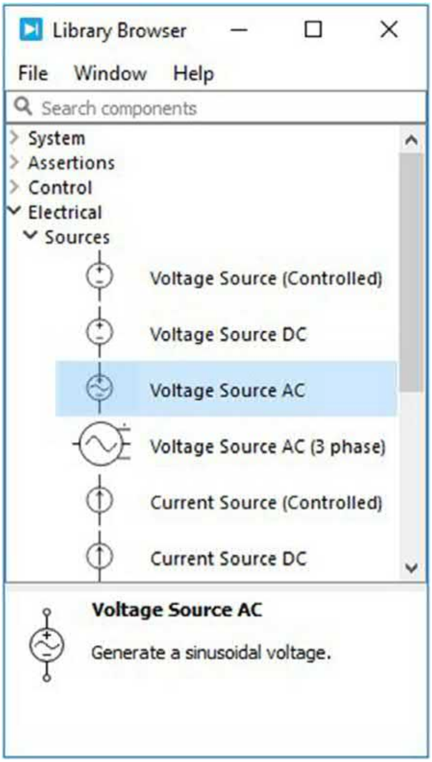

RCWithACinput
File
Edit
View
Simulation
Format
Window
Help
R1
R: 10
V_ac
C1
C: 100e-6

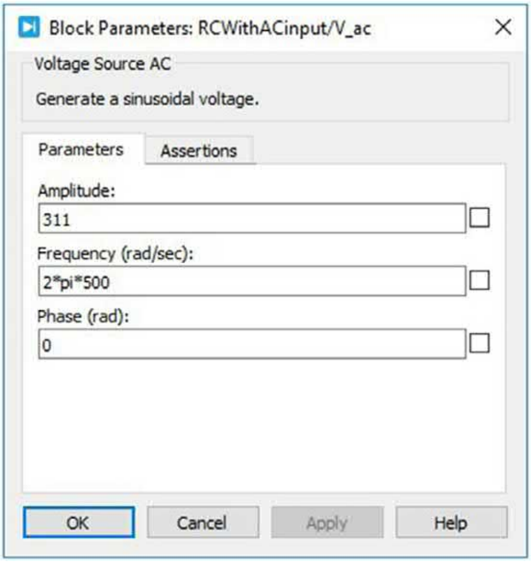
FIG. 2.95 RC circuit with AC source.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Voltage Source (Controlled)
Voltage Source DC
Voltage Source AC
Voltage Source AC (3 phase)
Current Source (Controlled)
Current Source DC

Voltage Source AC

Generate a sinusoidal voltage.

FIG. 2.96 AC voltage source block.


Block Parameters: RCWithACinput/V_ac

Voltage Source AC

Generate a sinusoidal voltage.

Amplitude: 311 ☐

Frequency (rad/sec): 2*pi*500 ☐

Phase (rad): 0 ☐
Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 2.97 Settings of the AC source block in Fig. 2.95.

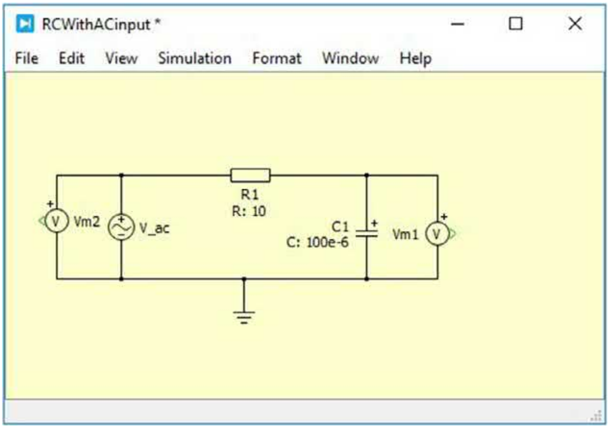
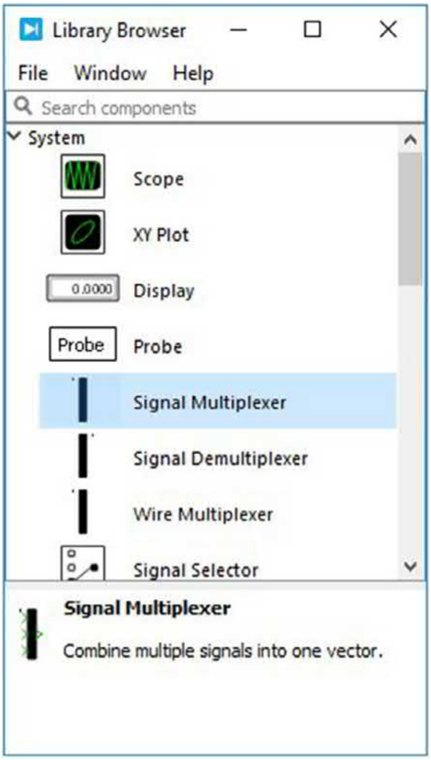

RCWithACinput *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 10
C1
C: 100e-6
Vm2
V_ac
Vm1
V
V
+
+
+
+

FIG. 2.98 Adding 2 voltmeter blocks to the schematic.

2.2.4 Multiple input scope

You can display different waveform using a scope with multiple axis. Assume the circuit shown in Fig. 2.105. We want to show output of ammeter and voltmeter on different axis.

Double click on scope block. The window shown in Fig. 2.106 will appear.


Library Browser
File
Window
Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
Signal Multiplexer

Combine multiple signals into one vector.

FIG. 2.99 Multiplexer block.

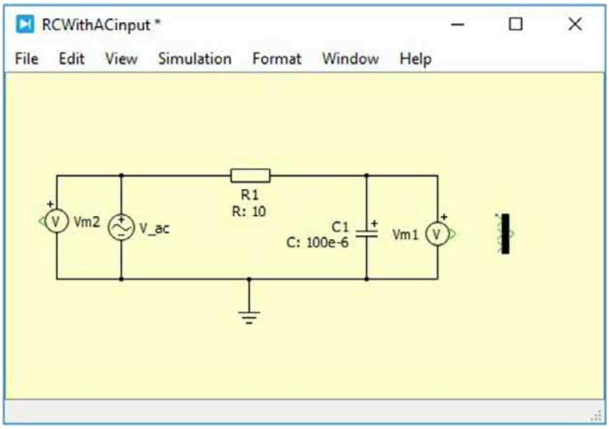
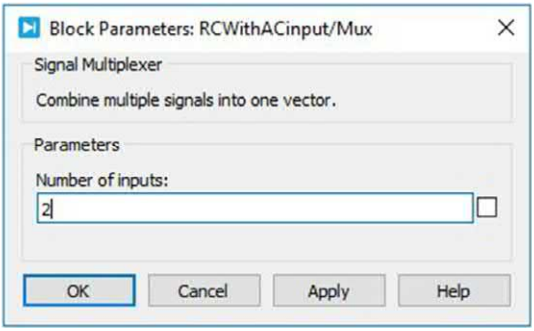

RCWithACinput *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 10
Vm2
V_ac
C1
C: 100e-6
Vm1
V
V

FIG. 2.100 Adding the multiplexer block to the schematic.



Block Parameters: RCWithACinput/Mux

Signal Multiplexer

Combine multiple signals into one vector.

Parameters

Number of inputs:

2

☐
OK
Cancel
Apply
Help

FIG. 2.101 Specifying the required number of inputs.

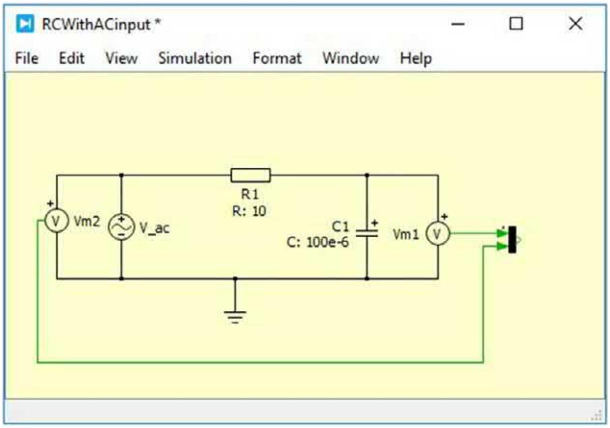

RCWithACinput *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 10
Vm2
V_ac
C1
C: 100e-6
Vm1
V
V

FIG. 2.102 Connecting the voltmeter outputs to the multiplexer block.

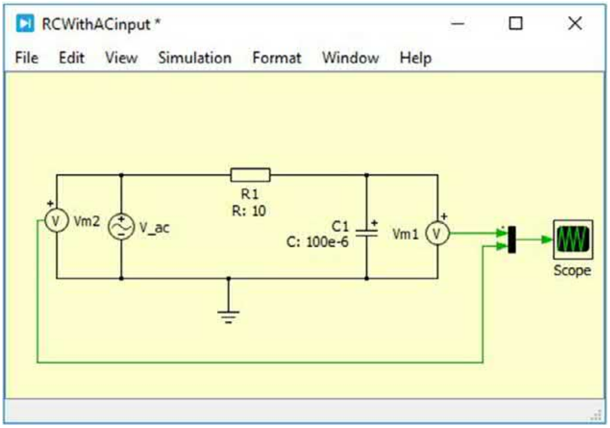

RCWithACinput *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 10
Vm2
V_ac
C1
C: 100e-6
Vm1
V
V
Scope

FIG. 2.103 Connecting the multiplexer block to the scope block.

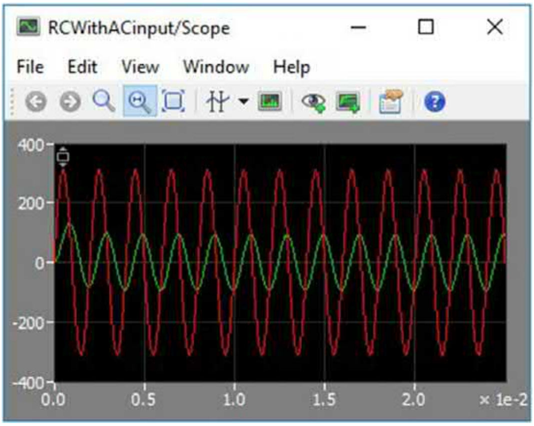

RCWithACinput/Scope
File Edit View Window Help
400
200
0
-200
-400
0.0
0.5
1.0
1.5
2.0
× 1e-2

FIG. 2.104 Simulation result (input source voltage and capacitor voltage).

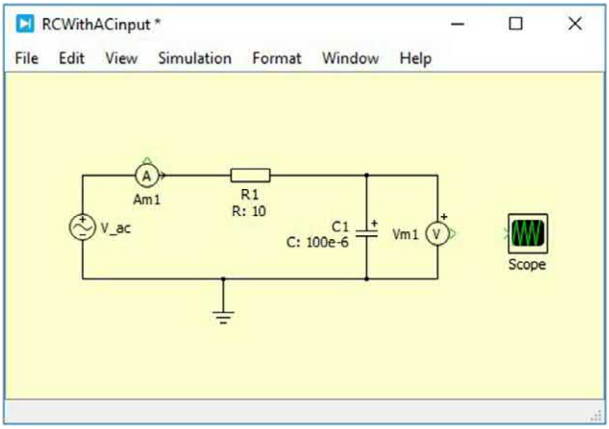

RCWithACinput *
File Edit View Simulation Format Window Help
A
Am1
R1
R: 10
V_ac
C1
C: 100e-6
Vm1
V
Scope

FIG. 2.105 Scope block can be turned into a multiple input block.

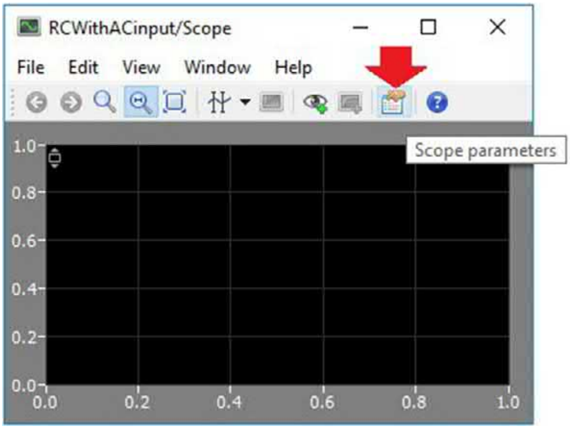
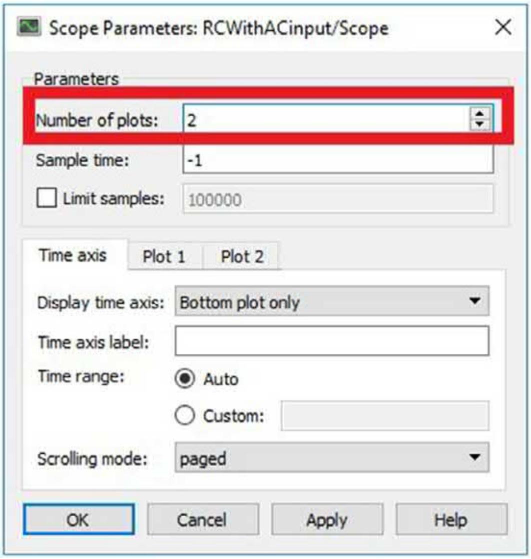

RCWithACinput/Scope
File Edit View Window Help
Scope parameters
1.0
0.8
0.6
0.4
0.2
0.0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 2.106 Scope parameters icon.



Scope Parameters: RCWithACinput/Scope

Parameters

Number of plots: 2

Sample time: -1

☐ Limit samples: 100000

Display time axis: Bottom plot only

Time axis label:

Time range: ● Auto ○ Custom:

Scrolling mode: paged

Time axis
Plot 1
Plot 2
OK
Cancel
Apply
Help

FIG. 2.107 Scope parameters window.

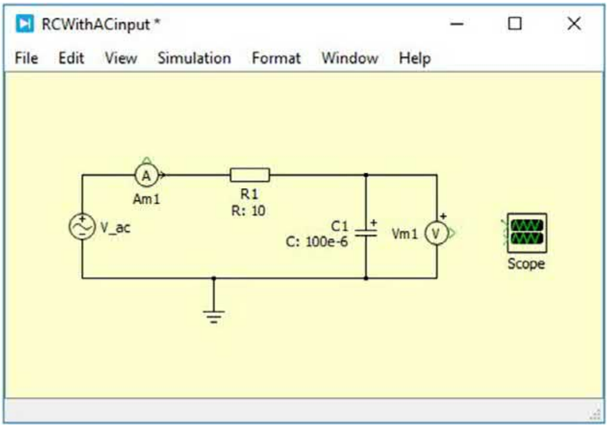

RCWithACinput *
File
Edit
View
Simulation
Format
Window
Help
A
Am1
R1
R: 10
V_ac
C1
C: 100e-6
Vm1
V
Scope

FIG. 2.108 Scope turned into a multiple input block.

Click on "Scope parameters" icon (see Fig. 2.106). After clicking the icon, window shown in Fig. 2.107 will appear. Increase the "Number of plots:" to 2 and click "OK" button.

The scope has 2 inputs now (Fig. 2.108).

Connect the ammeter and voltmeter to the scope (Fig. 2.109).

If we run the simulation with settings shown in Fig. 2.110, we obtain the waveforms shown in Fig. 2.111.

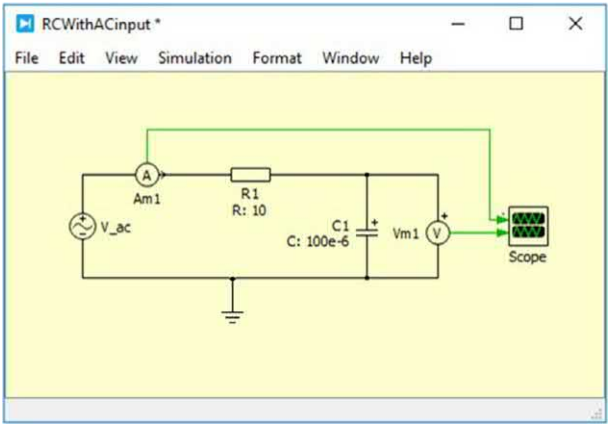
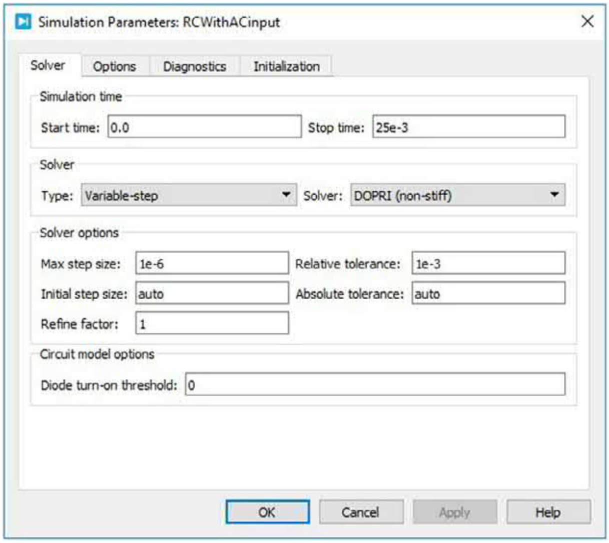

RCWithACinput *
File
Edit
View
Simulation
Format
Window
Help
A
Am1
R1
R: 10
V_ac
C1
C: 100e-6
Vm1
V
Scope

FIG. 2.109 Connecting the measurement outputs to the scope.


Simulation Parameters: RCWithACinput
Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

Simulation time

Start time: 0.0

Stop time: 25e-3

Solver

Type: Variable-step

Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-6

Relative tolerance: 1e-3

Initial step size: auto

Absolute tolerance: auto

Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

FIG. 2.110 Simulation parameters window.

2.2.5 XY scope block

The scope block shows a waveform in terms of time, i.e., $f(t)$. You can show a waveform in terms of another waveform. Using this capability, you can see the Lissajous curves. Consider the schematic shown in Fig. 2.112. We want to draw the capacitor voltage in terms of input source voltage.

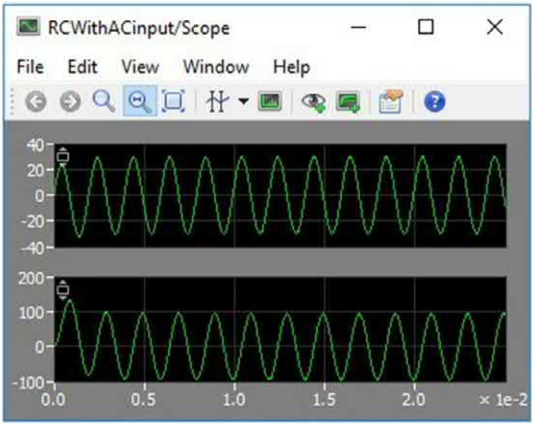

RCWithACinput/Scope
File Edit View Window Help
40
20
0
-20
-40
200
100
0
-100
0.0
0.5
1.0
1.5
2.0
× 1e-2

FIG. 2.111 Simulation result.

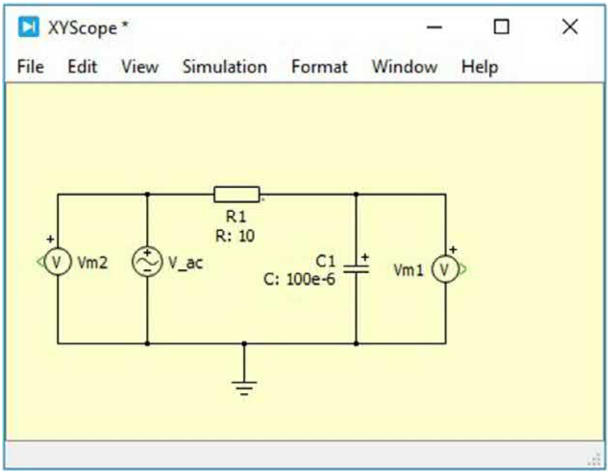

XYScope *
File Edit View Simulation Format Window Help
R1
R: 10
Vm2
V_ac
C1
C: 100e-6
Vm1

FIG. 2.112 You can show a waveform in terms of another waveform using XY Plot block.

Add an “XY Plot” block to the schematic. You can find the “XY Plot” block in “System” section of Library Browser (Figs. 2.113 and 2.114).

“XY Plot” block has two inputs. Lower input (Y axis) is drawn in terms of upper input (X axis) (Fig. 2.115).

Connect the voltmeters to the “XY Plot” block (Fig. 2.116).

If we run the simulation with settings shown in Fig. 2.117, we obtain the waveforms shown in Fig. 2.118.

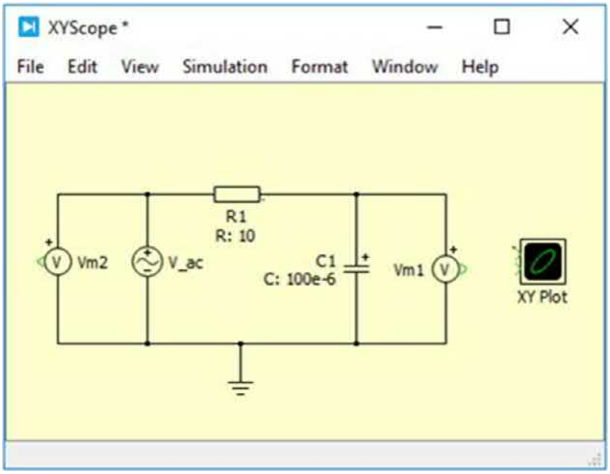
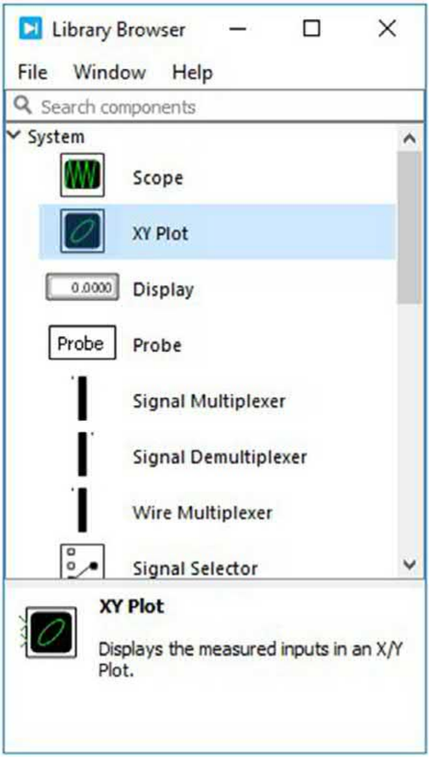

XYScope *
File Edit View Simulation Format Window Help
R1
R: 10
Vm2
V_ac
C1
C: 100e-6
Vm1
XY Plot

FIG. 2.113 Adding the XY Plot block to the schematic.


Library Browser
File Window Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector

XY Plot

Displays the measured inputs in an X/Y Plot.

FIG. 2.114 XY Plot block.

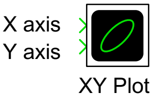

X axis
Y axis
XY Plot

FIG. 2.115 XY Plot block inputs. Upper one controls the X axis and the lower one controls the Y axis.

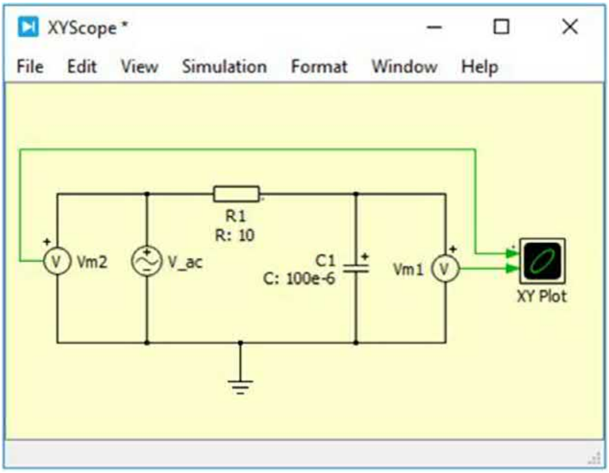
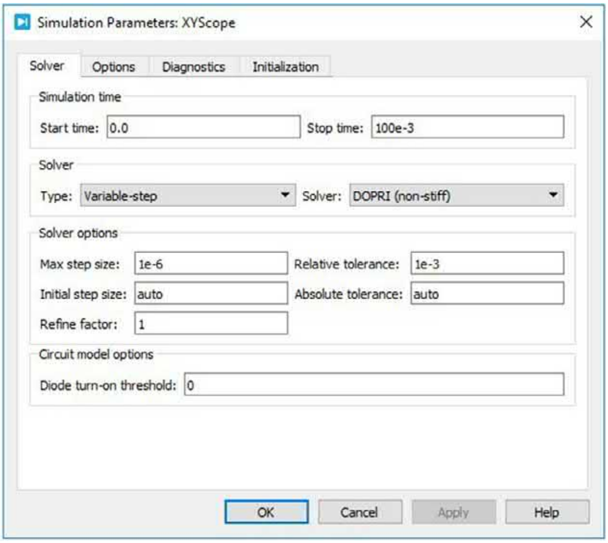

XYScope *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 10
V
Vm2
V_ac
C1
C: 100e-6
Vm1
V
XY Plot

FIG. 2.116 Connecting the output of measurement devices to the XY Plot block.


Simulation Parameters: XYScope
Solver
Options
Diagnostics
Initialization

Simulation time

Start time: 0.0

Stop time: 100e-3

Solver

Type: Variable-step

Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-6

Relative tolerance: 1e-3

Initial step size: auto

Absolute tolerance: auto

Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

OK
Cancel
Apply
Help

FIG. 2.117 Simulation settings.

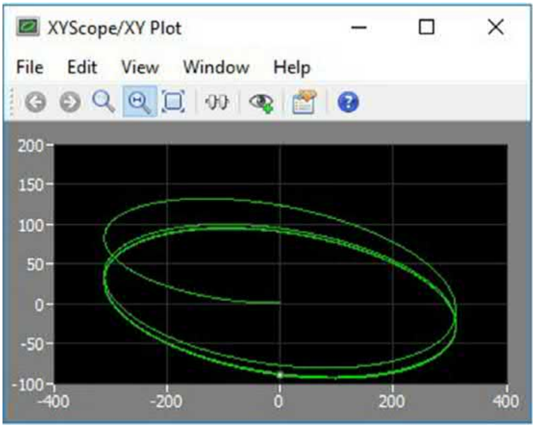

XYScope/XY Plot
File
Edit
View
Window
Help
200
150
100
50
0
-50
-100
-400
-200
0
200
400

FIG. 2.118 Simulation result.

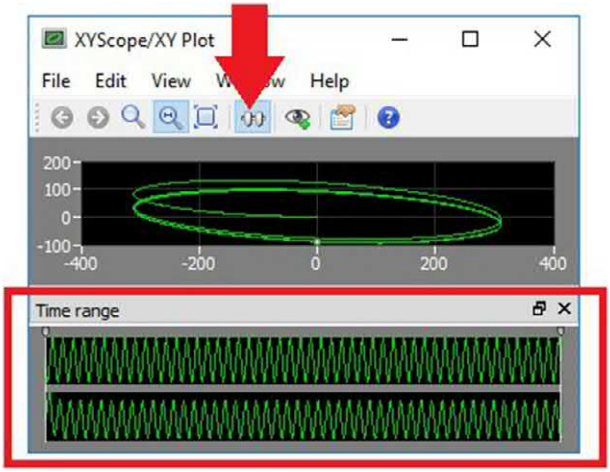

XYScope/XY Plot
File
Edit
View
Help
200
100
0
-100
-400
-200
0
200
400
Time range

FIG. 2.119 "Time range" icon.

If you want to see the steady-state curve, only click on the "Time range" icon. The "Time range" section will appear (Fig. 2.119).

Bring the appeared left sliders to the steady-state parts of the graph, i.e., toward right (Figs. 2.120 and 2.121).

The steady-state Lissajous curve will appear as shown in Fig. 2.122.

You can copy the curve into the Windows clipboard by clicking the "Copy…" (Fig. 2.123).

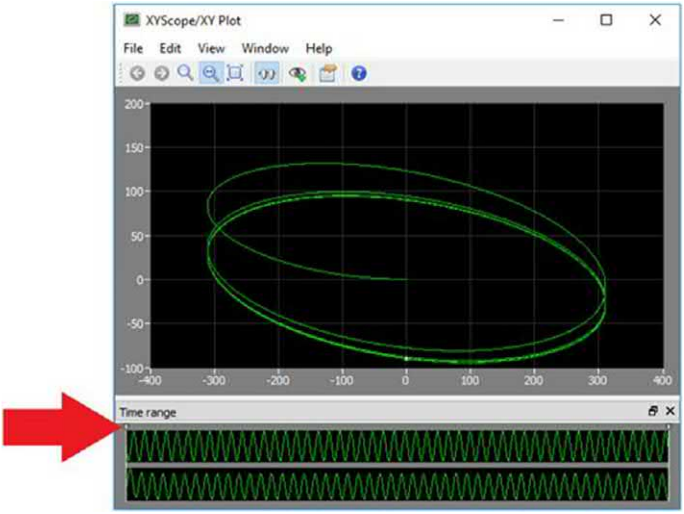

XYScope/XY Plot
File Edit View Window Help
200
150
100
50
0
-50
-100
-400
-300
-200
-100
0
100
200
300
400
Time range

FIG. 2.120 Use the slider to see only the steady-state part of graph.

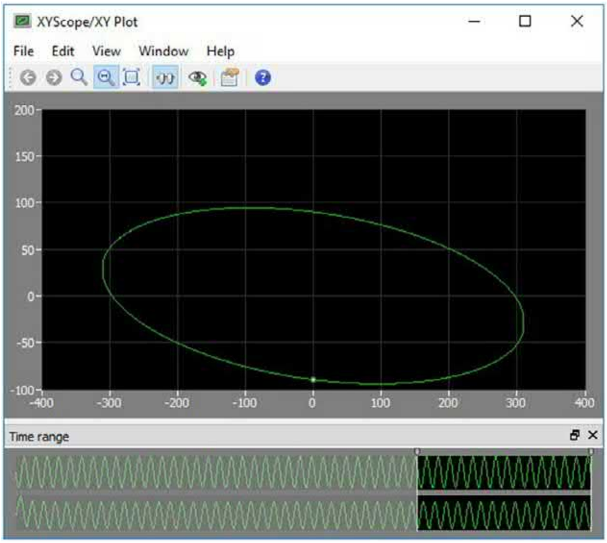

XYScope/XY Plot
File Edit View Window Help
200
150
100
50
0
-50
-100
-400
-300
-200
-100
0
100
200
300
400
Time range

FIG. 2.121 Move the left slider toward right to obtain the steady-state Lissajous curve.

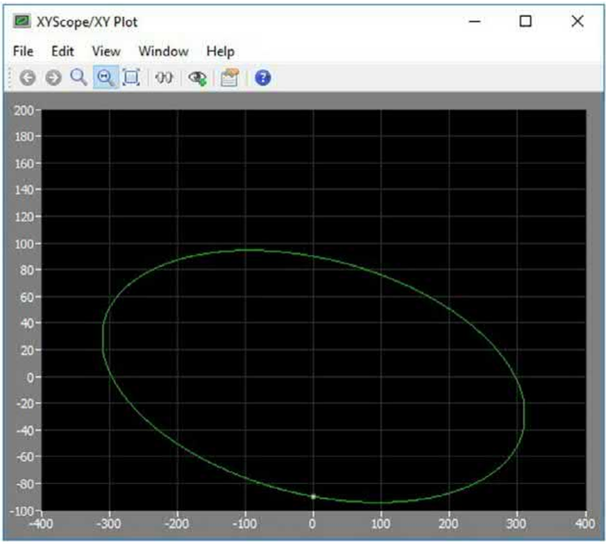

XYScope/XY Plot
File Edit View Window Help
200
180
160
140
120
100
80
60
40
20
0
-20
-40
-60
-80
-100
-400
-300
-200
-100
0
100
200
300
400

FIG. 2.122 Steady-state Lissajous curve.

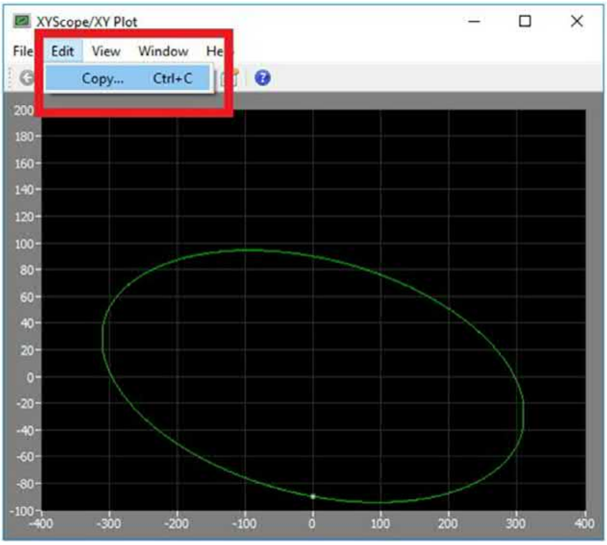
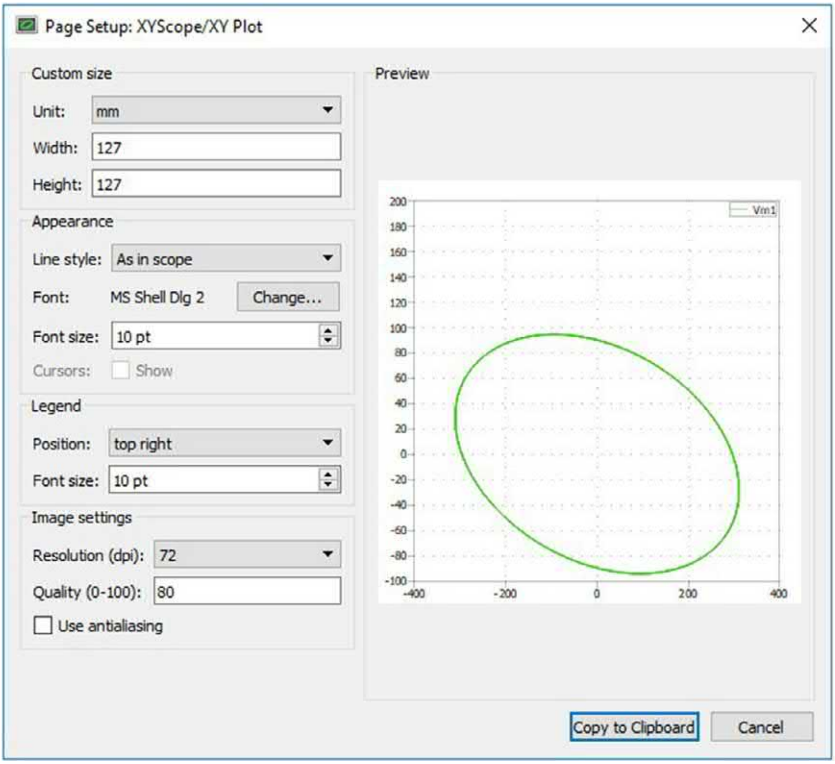

XYScope/XY Plot
File Edit View Window He
Copy... Ctrl+C
200
180
160
140
120
100
80
60
40
20
0
-20
-40
-60
-80
-100
-400
-300
-200
-100
0
100
200
300
400

FIG. 2.123 Copying the obtained graph into the Windows clipboard.



Page Setup: XYScope/XY Plot

Custom size
Unit: mm
Width: 127
Height: 127

Appearance
Line style: As in scope
Font: MS Shell Dlg 2
Font size: 10 pt
Cursors: ☐ Show

Legend
Position: top right
Font size: 10 pt

Image settings
Resolution (dpi): 72
Quality (0-100): 80
☐ Use antialiasing

Preview
Vm1
200
180
160
140
120
100
80
60
40
20
0
-20
-40
-60
-80
-100
-400
-200
0
200
400
Change...
Copy to Clipboard
Cancel

FIG. 2.124 Page setup window.

After clicking the "Edit…" the window shown in Fig. 2.124 appears, which can be used to obtain the desired image settings.

2.2.6 Simulation of control systems

PLECS can simulate both continuous time and discrete time control systems. Related blocks are shown in Figs. 2.125 and 2.126.

Assume that we want to simulate the block diagram shown in Fig. 2.127. Place of blocks required for this system is shown in Figs. 2.128–2.130.

Draw the schematic shown in Fig. 2.131.

Double click on the "Step" block and do the setting as shown in Fig. 2.132.

In order to obtain the $H(s) = \frac{1}{s^3 + 6s^2 + 5s}$, the "Plant" block settings must be done as shown in Fig. 2.133.

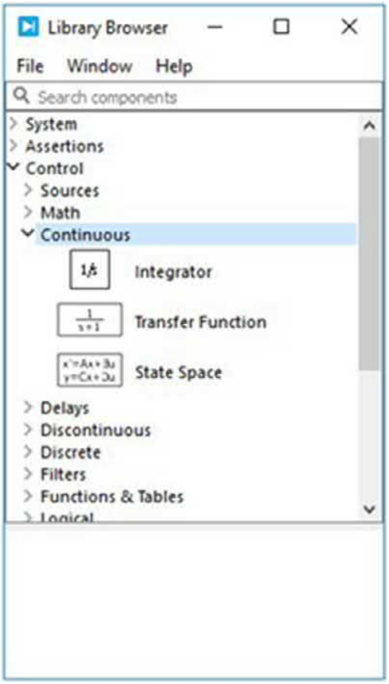

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
1/s
Integrator
Transfer Function
State Space
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical

FIG. 2.125 Continuous time blocks.

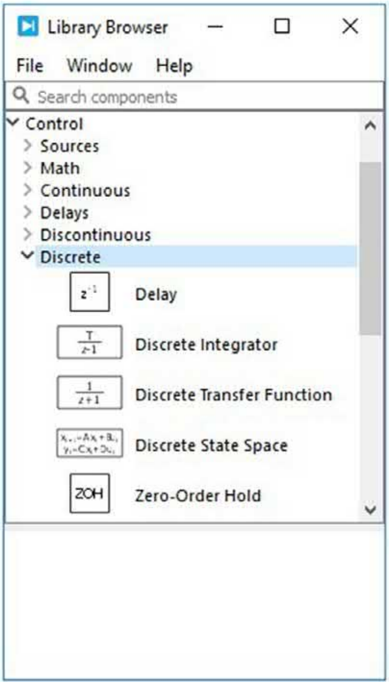

Library Browser
File
Window
Help
Search components
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Delay
Discrete Integrator
Discrete Transfer Function
Discrete State Space
ZOH
Zero-Order Hold

FIG. 2.126 Discrete time blocks.

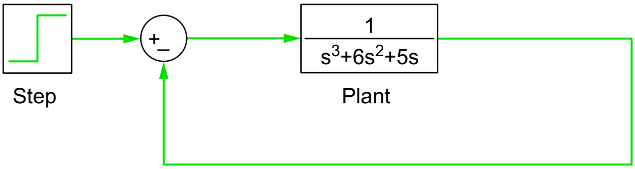
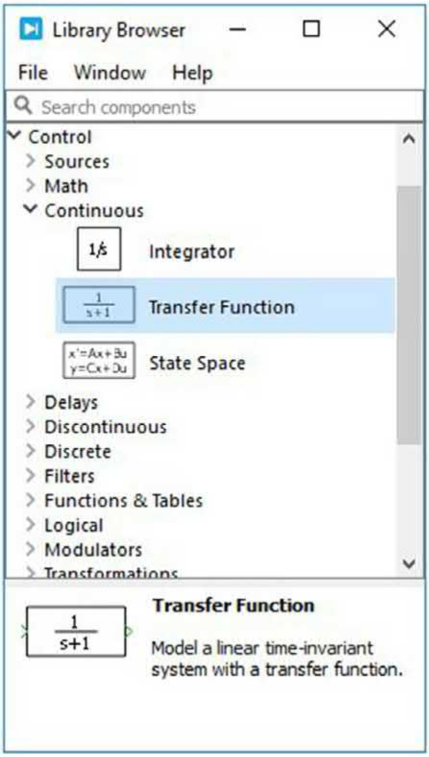

Step
+
_
1
s3+6s2+5s
Plant

FIG. 2.127 A simple closed-loop system.


Library Browser
File
Window
Help
Search components
Control
Sources
Math
Continuous
1/s
Integrator
1
s+1
Transfer Function
x'=Ax+Bu
y=Cx+Du
State Space
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Transformations
1
s+1

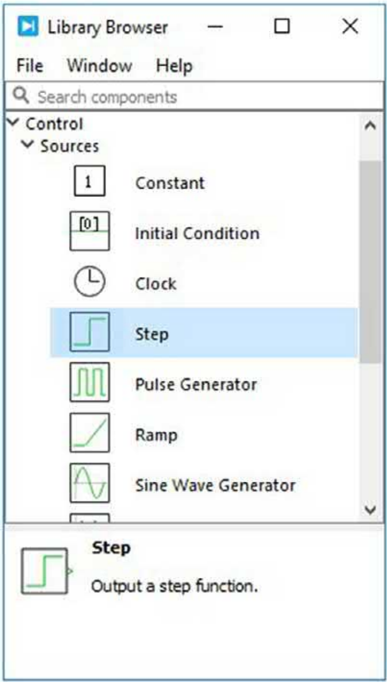
Transfer Function

Model a linear time-invariant system with a transfer function.

FIG. 2.128 Transfer function block.

After setting the “Plant” coefficients, the schematic changes to that shown in Fig. 2.134.

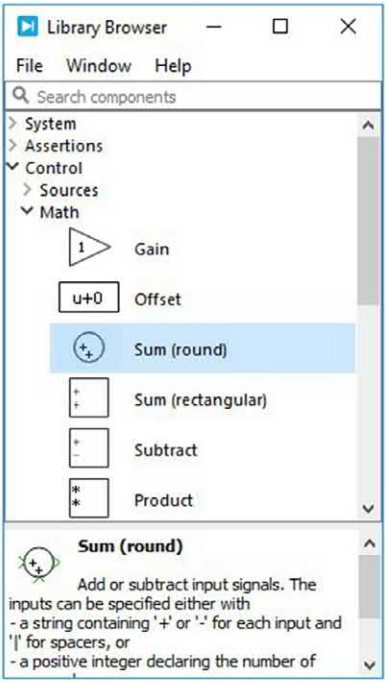
If we run the simulation with settings shown in Fig. 2.135, we obtain the waveform shown in Fig. 2.136.


Library Browser
File
Window
Help
Search components
Control
Sources
1
Constant
[0]
Initial Condition
Clock
Step
Pulse Generator
Ramp
Sine Wave Generator

Step

Output a step function.

FIG. 2.129 Step block.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
1
Gain
u+0
Offset
Sum (round)
Sum (rectangular)
Subtract
*
*
Product

Sum (round)

Add or subtract input signals. The inputs can be specified either with
- a string containing '+' or '-' for each input and '|' for spacers, or
- a positive integer declaring the number of

FIG. 2.130 Summation block.

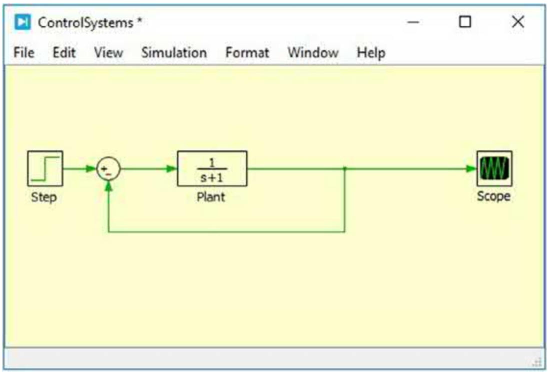
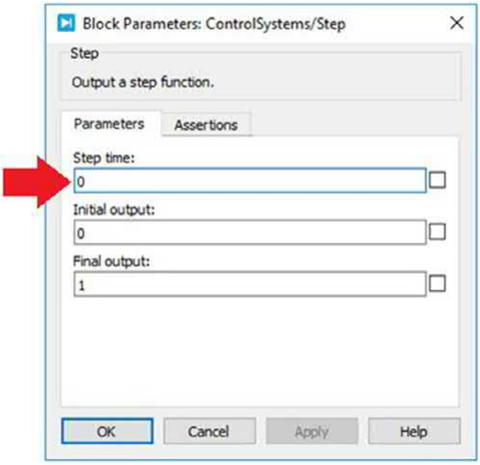

ControlSystems *
File
Edit
View
Simulation
Format
Window
Help
+
-
Step
$\frac{1}{s+1}$
Plant
Scope

FIG. 2.131 Drawing the close loop system.


Block Parameters: ControlSystems/Step

Step

Output a step function.

Parameters
Assertions

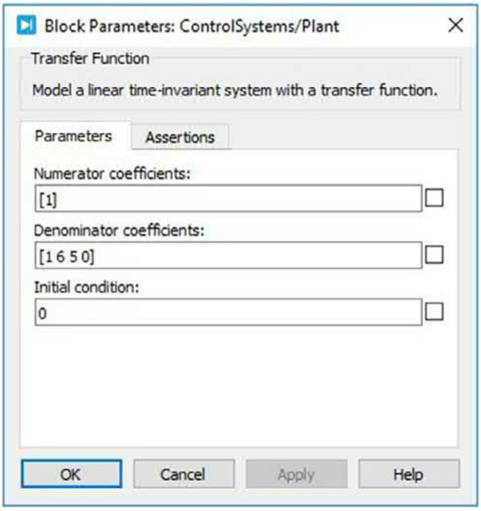
Step time: 0 ☐

Initial output: 0 ☐

Final output: 1 ☐

OK
Cancel
Apply
Help

FIG. 2.132 Step block settings.


Block Parameters: ControlSystems/Plant

Transfer Function

Model a linear time-invariant system with a transfer function.

Parameters
Assertions

Numerator coefficients: [1] ☐

Denominator coefficients: [1 6 5 0] ☐

Initial condition: 0 ☐

OK
Cancel
Apply
Help

FIG. 2.133 Plant block settings.

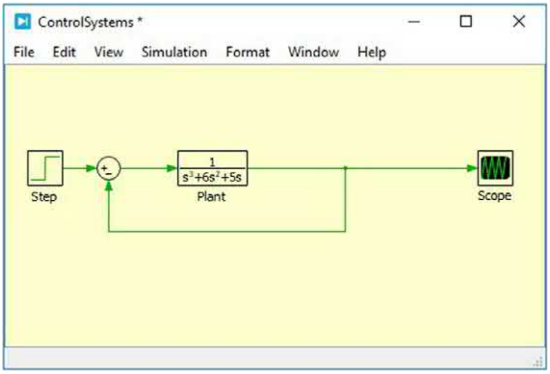
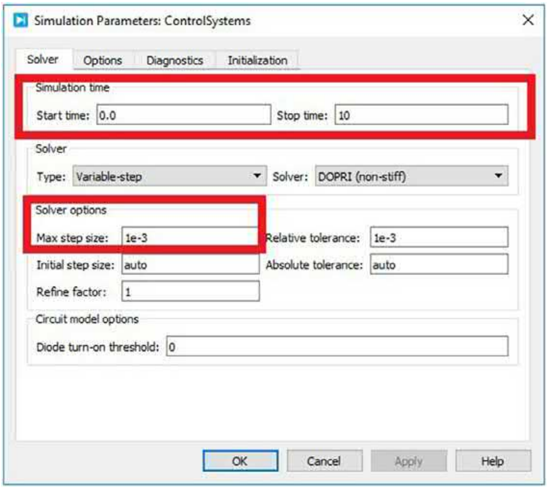

ControlSystems *
File
Edit
View
Simulation
Format
Window
Help
Step
$\frac{1}{s^3+6s^2+5s}$
Plant
Scope

FIG. 2.134 Ready-to-use simulation schematic.


Simulation Parameters: ControlSystems
Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

Simulation time
Start time: 0.0
Stop time: 10

Solver
Type: Variable-step
Solver: DOPRI (non-stiff)

Solver options
Max step size: 1e-3
Relative tolerance: 1e-3
Initial step size: auto
Absolute tolerance: auto
Refine factor: 1

Circuit model options
Diode turn-on threshold: 0

FIG. 2.135 Simulation settings.

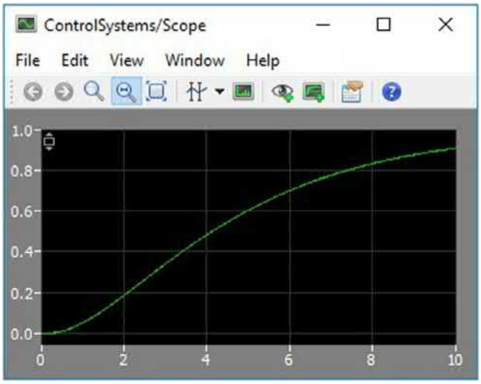

ControlSystems/Scope
File
Edit
View
Window
Help
1.0
0.8
0.6
0.4
0.2
0.0
0
2
4
6
8
10

FIG. 2.136 Simulation result.

2.2.7 Getting help in PLECS

PLECS has a powerful help system. For example, assume that you need some information about a block you placed in the schematic (Fig. 2.137). Simply right click on the block and select the "Help."

The block information window will appear (Fig. 2.138).

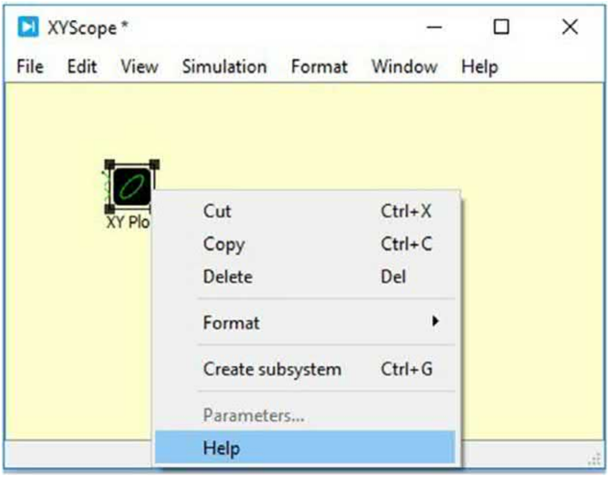
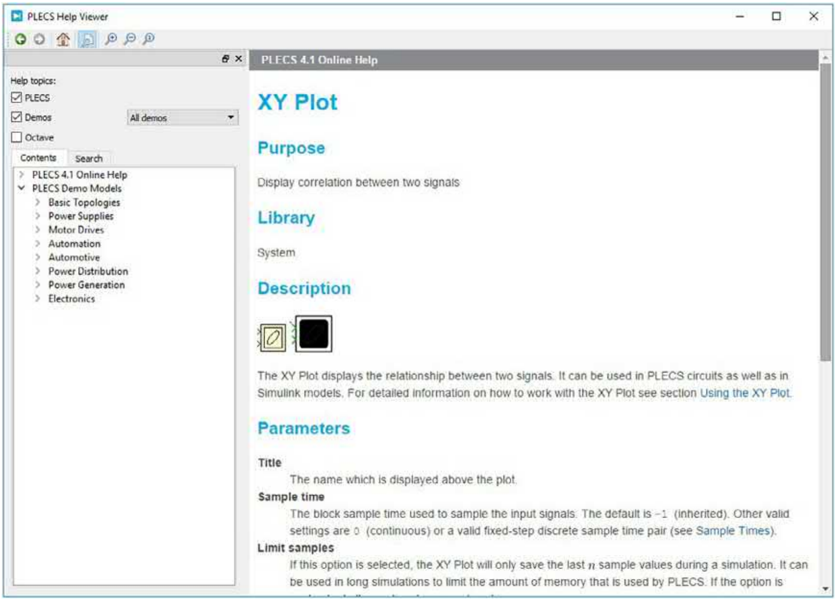

XYScope *
File
Edit
View
Simulation
Format
Window
Help
XY Plo
Cut Ctrl+X
Copy Ctrl+C
Delete Del
Format
Create subsystem Ctrl+G
Parameters...
Help

FIG. 2.137 You can see a block documentation by right clicking on it and selecting the Help.


PLECS Help Viewer
Help topics:
☑ PLECS
☑ Demos
All demos
☐ Octave
Contents
Search
PLECS 4.1 Online Help
PLECS Demo Models
Basic Topologies
Power Supplies
Motor Drives
Automation
Automotive
Power Distribution
Power Generation
Electronics
PLECS 4.1 Online Help

XY Plot

Purpose

Display correlation between two signals

Library

System

Description

The XY Plot displays the relationship between two signals. It can be used in PLECS circuits as well as in Simulink models. For detailed information on how to work with the XY Plot see section Using the XY Plot.

Parameters

Title
The name which is displayed above the plot.

Sample time
The block sample time used to sample the input signals. The default is -1 (inherited). Other valid settings are 0 (continuous) or a valid fixed-step discrete sample time pair (see Sample Times).

Limit samples
If this option is selected, the XY Plot will only save the last n sample values during a simulation. It can be used in long simulations to limit the amount of memory that is used by PLECS. If the option is

FIG. 2.138 Online Help for XY Plot block.

You can access the help system by clicking the "PLECS Documentation" as well (Fig. 2.139).

You can access to plenty of sample simulations by clicking the menus in the left of the appeared window (Fig. 2.140).

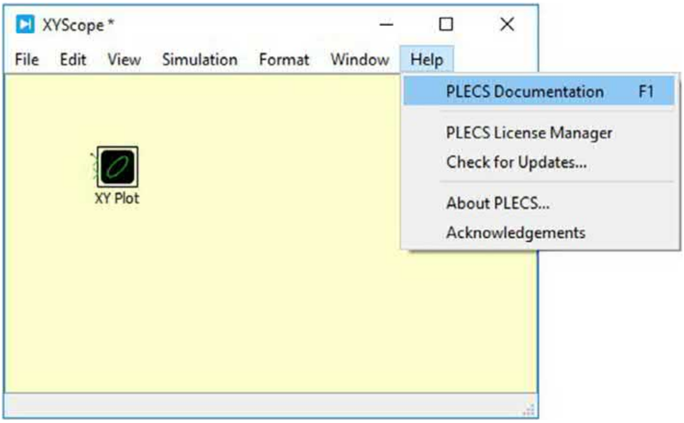
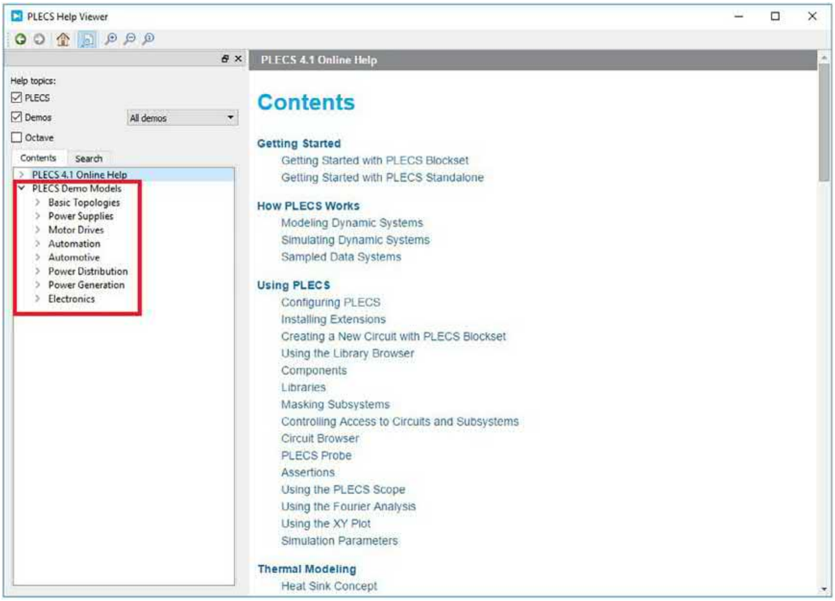

XYScope *
File
Edit
View
Simulation
Format
Window
Help
PLECS Documentation F1
PLECS License Manager
Check for Updates...
About PLECS...
Acknowledgements
XY Plot

FIG. 2.139 PLECS documentation.


PLECS Help Viewer
Help topics:
☑ PLECS
☑ Demos
All demos
☐ Octave
Contents
Search
PLECS 4.1 Online Help
PLECS Demo Models
Basic Topologies
Power Supplies
Motor Drives
Automation
Automotive
Power Distribution
Power Generation
Electronics
PLECS 4.1 Online Help

Contents

Getting Started
- Getting Started with PLECS Blockset
- Getting Started with PLECS Standalone

How PLECS Works
- Modeling Dynamic Systems
- Simulating Dynamic Systems
- Sampled Data Systems

Using PLECS
- Configuring PLECS
- Installing Extensions
- Creating a New Circuit with PLECS Blockset
- Using the Library Browser
- Components
- Libraries
- Masking Subsystems
- Controlling Access to Circuits and Subsystems
- Circuit Browser
- PLECS Probe
- Assertions
- Using the PLECS Scope
- Using the Fourier Analysis
- Using the XY Plot
- Simulation Parameters

Thermal Modeling
- Heat Sink Concept

FIG. 2.140 Demo models provided by Plexim can be a good reference to learn the art of simulation.

References

[1] PLECS user manual available online at, https://www.plexim.com/download/documentation.

Further reading

[2] D. Maksimovic, A.M. Stankovic, V.J. Thottuvelil, G.C. Verghese, Modeling and simulation of power electronic converters, Proc. IEEE 89 (6) (Jun 2001) 898–912, https://doi.org/10.1109/5.931486.
[3] M.H. Rashid, SPICE for Power Electronics and Electric Power, CRC Press, 2017.
[4] F. Asadi, N. Abut, Simulation of power electronics converters with PSIM®, (in Turkish), Umuttepe yayınları, 2018.
[5] F. Asadi, N. Abut, Simulation of power electronics converters with MATLAB/Simulink®, (in Turkish), Umuttepe yayınları, 2018.

Chapter 3

Basics of power electronic circuits simulation with PLECS

Chapter outline

Introduction **76**

3.1 Example 3.1: MOSFET with resistive load **76**

- 3.1.1 Preparing the simulation 76
- 3.1.2 Measuring the average and RMS of waveforms 82
- 3.1.3 Measuring the power dissipated in the load resistor 86
- 3.1.4 Subsystem block 89
- 3.1.5 Measuring the input power 97
- 3.1.6 Generating the PWM signal using ready-to-use blocks 101

3.2 Example 3.2: Uncontrolled single-phase half-wave rectifier **104**

- 3.2.1 Preparing the simulation 104
- 3.2.2 Harmonic content of output 107
- 3.2.3 Measuring the RMS values of voltages/currents 112
- 3.2.4 Capturing a period of output voltage/current 114
- 3.2.5 “Discrete RMS value” block 116
- 3.2.6 “Discrete mean value” block 120
- 3.2.7 Measuring the maximum/minimum of waveforms shown in the scope block 122
- 3.2.8 Obtaining the load instantaneous power 124

3.3 Example 3.3: Single-phase half-wave controlled rectifier **127**

- 3.3.1 Preparing the simulation 128
- 3.3.2 Calculating the RMS, mean, max/min, etc. 131

3.4 Example 3.4: Single-phase full-wave controlled rectifier **131**

- 3.4.1 Preparing the simulation 132
- 3.4.2 Calculating the average output voltage using the “Discrete Fourier transform” block 138

3.5 Example 3.5: 3 Phase full-wave controlled rectifier **142**

- 3.5.1 Preparing the simulation 142
- 3.5.2 Drawing more understandable schematics using “Electrical label,” “Signal from,” and “Signal goto” blocks 145
- 3.5.3 Delay block 150

3.6 Example 3.6: Boost converter **152**

- 3.6.1 Preparing the simulation 152
- 3.6.2 Simulating the circuit using the ready-to-use modulator 157
- 3.6.3 Efficiency measurement 159

Simulation of Power Electronics Converters Using PLECS®. https://doi.org/10.1016/B978-0-12-817364-0.00003-5

© 2020 Elsevier Inc. All rights reserved.

3.7 Example 3.7: Obtaining the small signal transfer functions for a buck converter 164
3.7.1 Preparing the simulation 164
3.7.2 Comparison of different simulation results 172
3.7.3 Importing the simulation results into the MATLAB 178
3.8 Example 3.8: Mutual inductance 180
3.8.1 Preparing the simulation 180
3.8.2 Using parametric variables to specify the component values 186
3.9 Example 3.9: 3-Phase inverter 188
3.9.1 Preparing the simulation 188
3.9.2 Calculating the total harmonic distortion (THD) 197
3.9.3 "Fourier series" block 204
3.10 Example 3.10: Simulation of electrical machines 209
3.10.1 Preparing the simulation 209
3.10.2 Monitoring using the probe block 215
References 217
Further reading 217

Introduction

This chapter shows you how simulation of power converter can be done with the aid of PLECS standalone. We give 12 different examples in this chapter. It is a good idea to do some hand analysis and compare the hand analysis results with the ones provided by PLECS. Try to find any source of error between your result and PLECS output. This helps you to understand the power electronics concepts very well.

3.1 Example 3.1: MOSFET with resistive load

In this example, we will simulate a MOSFET driving a resistive load.

3.1.1 Preparing the simulation

Assume a simulation such as that shown in Fig. 3.1. The MOSFET block can be found under the "Power Semiconductors" section of Library Browser (Fig. 3.2).

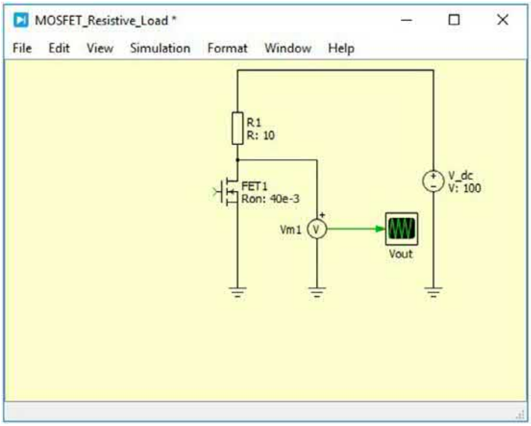
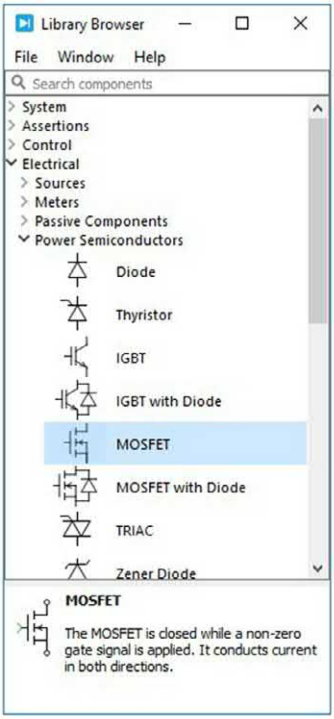


FIG. 3.1 MOSFET with resistive load.

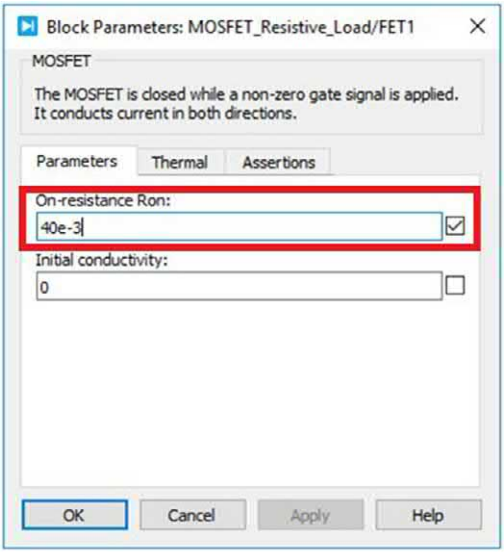

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Diode
Thyristor
IGBT
IGBT with Diode
MOSFET
MOSFET with Diode
TRIAC
Zener Diode

MOSFET

The MOSFET is closed while a non-zero gate signal is applied. It conducts current in both directions.

FIG. 3.2 MOSFET block.

Assume that the drain-source resistance is 40 mΩ. With this assumption, double click the MOSFET block and do the settings as shown in Fig. 3.3.


Block Parameters: MOSFET_Resistive_Load/FET1
Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

MOSFET

The MOSFET is closed while a non-zero gate signal is applied. It conducts current in both directions.

On-resistance Ron: 40e-3 ☑

Initial conductivity: 0 ☐

FIG. 3.3 Specifying the On-resistance of the MOSFET.

Assume that the MOSFET is triggered with the pulse shown in Fig. 3.4.

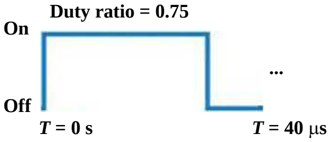

Duty ratio = 0.75
On
Off
...
T = 0 s
T = 40 µs

FIG. 3.4 The MOSFET gate signals.

To obtain the gate signal shown in Fig. 3.4, we change the schematic to that shown in Fig. 3.5. Added blocks places are shown in Figs. 3.6–3.8. "Triangular Wave" settings are shown in Fig. 3.9.

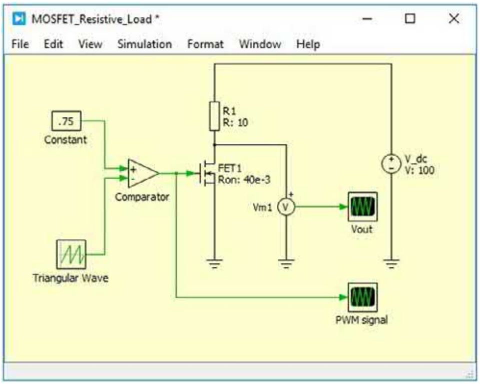
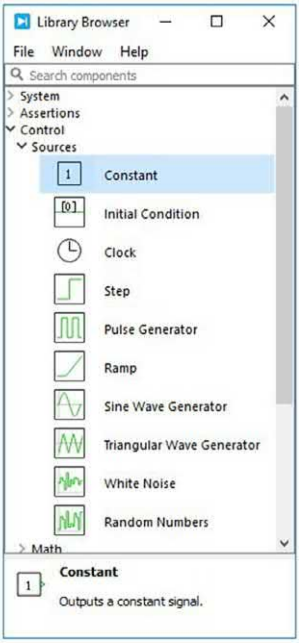

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
.75
Constant
Comparator
Triangular Wave
R1
R: 10
FET1
Ron: 40e-3
Vm1
V_dc
V: 100
Vout
PWM signal

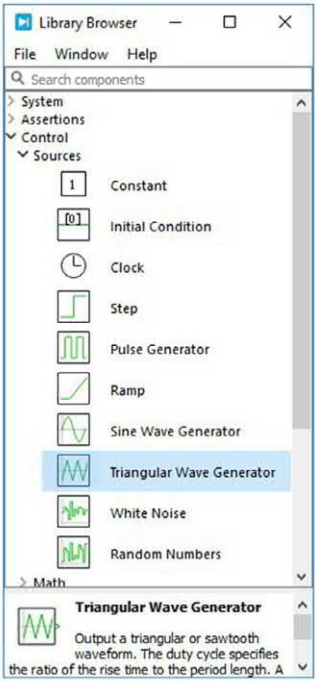
FIG. 3.5 Adding the PWM generation blocks to the schematic.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Constant
Initial Condition
Clock
Step
Pulse Generator
Ramp
Sine Wave Generator
Triangular Wave Generator
White Noise
Random Numbers
Math

Constant

Outputs a constant signal.

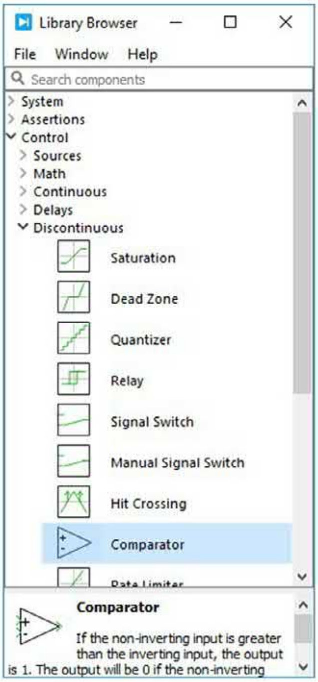
FIG. 3.6 Constant block.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Constant
Initial Condition
Clock
Step
Pulse Generator
Ramp
Sine Wave Generator
Triangular Wave Generator
White Noise
Random Numbers
Math

Triangular Wave Generator

Output a triangular or sawtooth waveform. The duty cycle specifies the ratio of the rise time to the period length. A

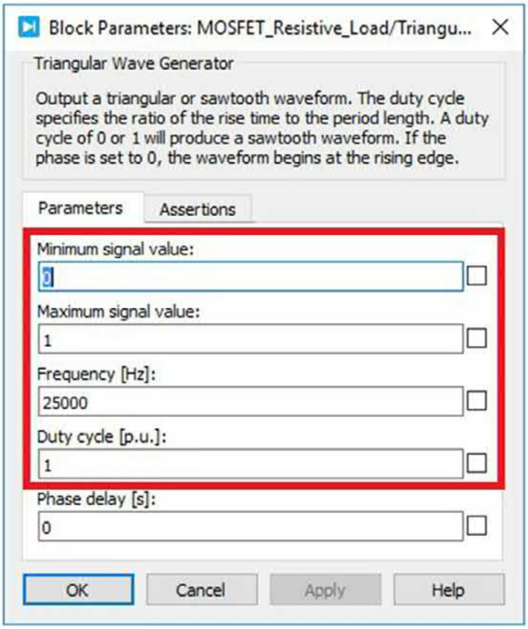
FIG. 3.7 Triangular Wave Generator block.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Saturation
Dead Zone
Quantizer
Relay
Signal Switch
Manual Signal Switch
Hit Crossing
Comparator
Rate Limiter

Comparator

If the non-inverting input is greater than the inverting input, the output is 1. The output will be 0 if the non-inverting

FIG. 3.8 Comparator block.



Triangular Wave Generator

Output a triangular or sawtooth waveform. The duty cycle specifies the ratio of the rise time to the period length. A duty cycle of 0 or 1 will produce a sawtooth waveform. If the phase is set to 0, the waveform begins at the rising edge.

Parameters | Assertions

Minimum signal value: 0 ☐

Maximum signal value: 1 ☐

Frequency [Hz]: 25000 ☐

Duty cycle [p.u.]: 1 ☐

Phase delay [s]: 0 ☐

FIG. 3.9 "Triangular Wave Generator" block settings.

Using the "Triangular Wave Generator" block, one can obtain saw tooth or triangular waveforms. See Figs. 3.10–3.12 for more information on this block setting.

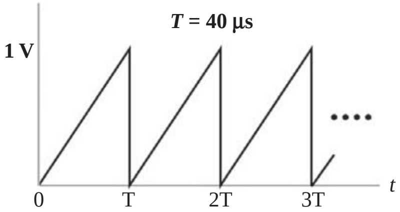


FIG. 3.10 Output of a "Triangular Wave Generator" block with Minimum signal value: 0, Maximum signal value: 1, Frequency: 25000, Duty cycle [p.u.]: 1.

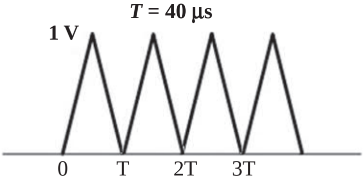


FIG. 3.11 Output of a "Triangular Wave Generator" block with Minimum signal value: 0, Maximum signal value: 1, Frequency: 25000, Duty cycle [p.u.]: 0.5.

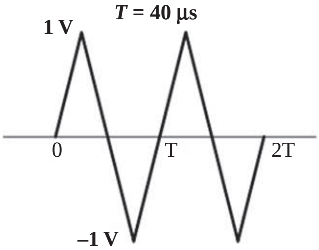
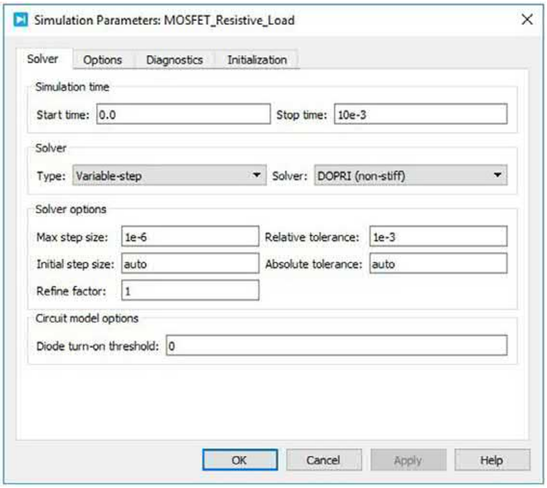

T = 40 μs
1 V
0
T
2T
–1 V

FIG. 3.12 Output of a “Triangular Wave Generator” block with Minimum signal value: −1, Maximum signal value: 1, Frequency: 25000, Duty cycle [p.u.]: 0.5.

If we run the simulation with the settings shown in Fig. 3.13, we obtain the waveform shown in Fig. 3.14.


Simulation Parameters: MOSFET_Resistive_Load

Solver
Options
Diagnostics
Initialization

Simulation time

Start time: 0.0
Stop time: 10e-3

Solver

Type: Variable-step
Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-6
Relative tolerance: 1e-3
Initial step size: auto
Absolute tolerance: auto
Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

OK
Cancel
Apply
Help

FIG. 3.13 Simulation parameters.

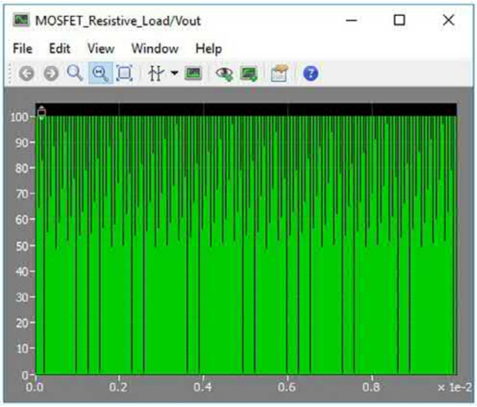


FIG. 3.14 Simulation result (“Vout” scope).

Use the magnifier icon in the toolbox to see the waveform better (Fig. 3.15).

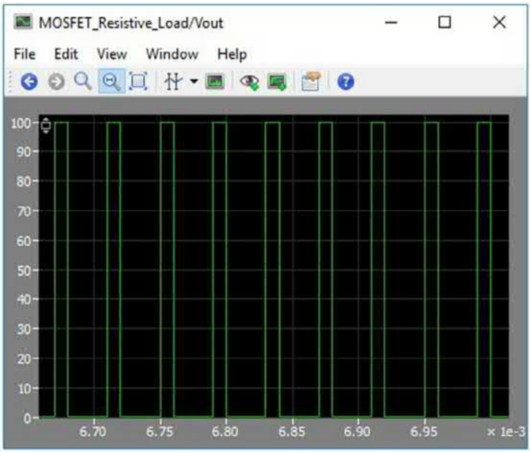

MOSFET_Resistive_Load/Vout
File Edit View Window Help
100
90
80
70
60
50
40
30
20
10
0
6.70
6.75
6.80
6.85
6.90
6.95
× 1e-3

FIG. 3.15 Load resistor voltage.

MOSFET gate signal is shown in Fig. 3.16. You can use the cursors to ensure that its duty ratio and frequency.

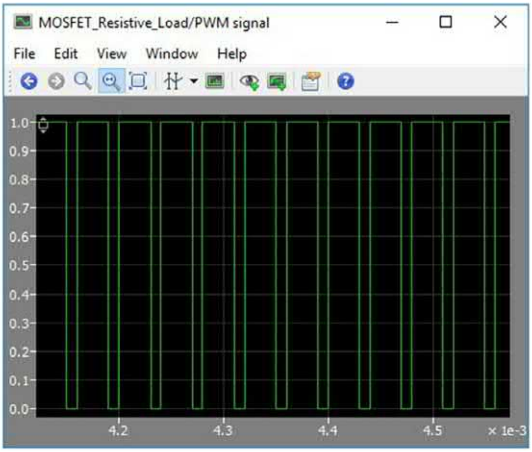
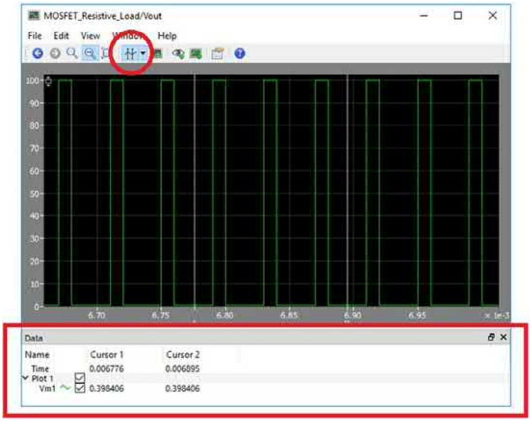

MOSFET_Resistive_Load/PWM signal
File Edit View Window Help
1.0
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0.0
4.2
4.3
4.4
4.5
× 1e-3

FIG. 3.16 MOSFET gate signal.

3.1.2 Measuring the average and RMS of waveforms

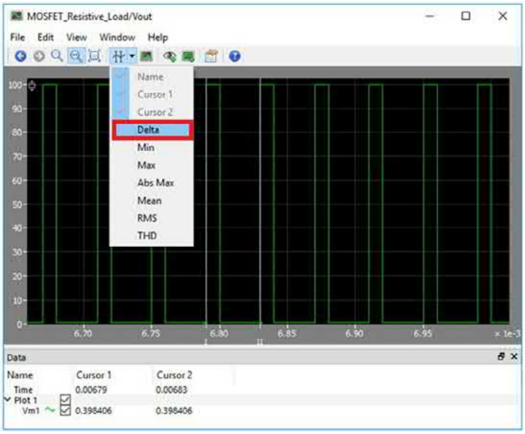
You can use the cursors to measure the average and Root Mean Square (RMS) value of output voltage. To do these measurements, click on Cursors icon (the icon shown with circle in Fig. 3.17). After clicking the Cursors icon, the “Data” section will appear (Fig. 3.17).


MOSFET_Resistive_Load/Vout
File
Edit
View
Window
Help
100
90
80
70
60
50
40
30
20
10
0
6.70
6.75
6.80
6.85
6.90
6.95
× 1e-3
Data

Name		Cursor 1	Cursor 2
Time		0.006776	0.006895
Plot 1	☑		
Vm1	☑	0.398406	0.398406

FIG. 3.17 Adding cursors to the scope.

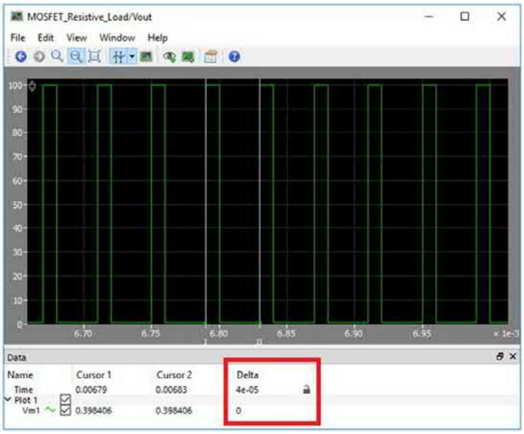
Click on the small triangle behind the icon. Click the “Delta” in appeared menu (Fig. 3.18).


MOSFET_Resistive_Load/Vout
File
Edit
View
Window
Help
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
RMS
THD
100
90
80
70
60
50
40
30
20
10
0
6.70
6.75
6.80
6.85
6.90
6.95
× 1e-3
Data

Name		Cursor 1	Cursor 2
Time		0.00679	0.00683
Plot 1	☑		
Vm1	☑	0.398406	0.398406

FIG. 3.18 Measuring the difference between the two cursors.

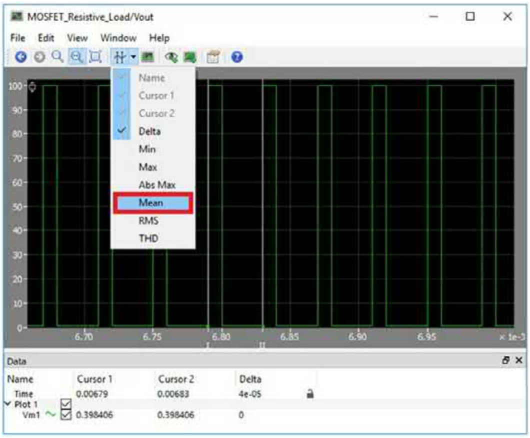
A section named “Delta” will be added to the “Data” section. Move the cursors until the time difference between them reaches to 40 μs. When the time difference reaches the 40 μs, you capture exactly one period of output voltage between the two cursors (Fig. 3.19).


MOSFET_Resistive_Load/Vout
File Edit View Window Help
100
90
80
70
60
50
40
30
20
10
0
6.70
6.75
6.80
6.85
6.90
6.95
× 1e-3
Data

Name	Cursor 1	Cursor 2	Delta
Time	0.00679	0.00683	4e-05
Plot 1			
Vm1	0.398406	0.398406	0

FIG. 3.19 Time difference between the two cursors is 4e − 5 = 4 × 10^{-5} = 40 μs and the voltage difference is 0 V.

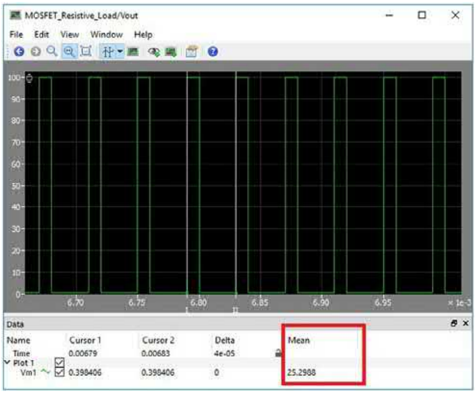
Click on the small triangle behind the icon and click the "Mean" in the appeared menu (Fig. 3.20).


MOSFET_Resistive_Load/Vout
File Edit View Window Help
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
RMS
THD
100
90
80
70
60
50
40
30
20
10
0
6.70
6.75
6.80
6.85
6.90
6.95
× 1e-3
Data

Name	Cursor 1	Cursor 2	Delta
Time	0.00679	0.00683	4e-05
Plot 1			
Vm1	0.398406	0.398406	0

FIG. 3.20 Measuring the mean value.

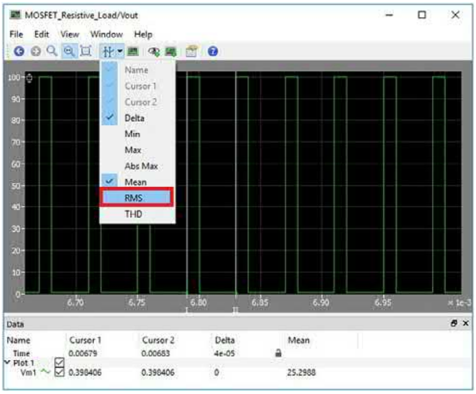
PLECS calculates and shows the average value of waveform captured between the two cursors (Fig. 3.21).


MOSFET_Resistive_Load/Vout
File Edit View Window Help
100
90
80
70
60
50
40
30
20
10
0
6.70
6.75
6.80
6.85
6.90
6.95
× 1e-3
Data

Name	Cursor 1	Cursor 2	Delta	Mean
Time	0.00679	0.00683	4e-05	
Plot 1				
Vm1	0.398406	0.398406	0	25.2988

FIG. 3.21 Load resistor average value is about 25.3 V.

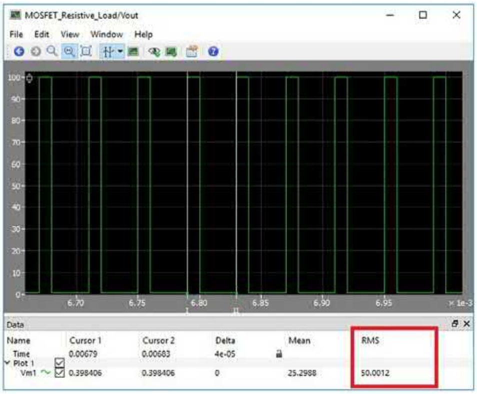
In order to calculate the RMS value of output voltage, click on the small triangle behind the ╫ icon and select the “RMS” in the appeared menu (Fig. 3.22).


MOSFET_Resistive_Load/Vout
File Edit View Window Help
Name
Cursor 1
Cursor 2
✓ Delta
Min
Max
Abs Max
✓ Mean
RMS
THD
100
90
80
70
60
50
40
30
20
10
0
6.70
6.75
6.80
6.85
6.90
6.95
× 1e-3
Data

Name	Cursor 1	Cursor 2	Delta	Mean
Time	0.00679	0.00683	4e-05	
Plot 1				
Vm1	0.398406	0.398406	0	25.2988

FIG. 3.22 Measuring the RMS.

PLECS calculates and shows the RMS value of output voltage (Fig. 3.23).

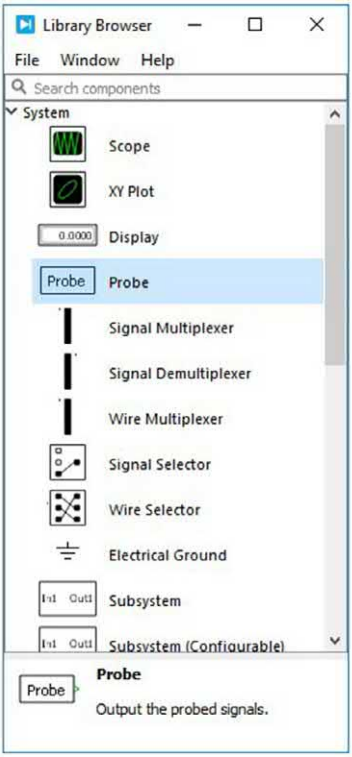

MOSFET_Resistive_Load/Vout
File Edit View Window Help
100
90
80
70
60
50
40
30
20
10
0
6.70
6.75
6.80
6.85
6.90
6.95
× 1e-3
Data

Name	Cursor 1	Cursor 2	Delta	Mean	RMS
Time	0.00679	0.00683	4e-05		
Plot 1					
Vm1	0.398406	0.398406	0	25.2988	50.0012

FIG. 3.23 Load resistor voltage has an RMS about 50 V.

3.1.3 Measuring the power dissipated in the load resistor

You can measure the dissipated power in the load resistor easily using the "Probe" block (Fig. 3.24).


Library Browser
File
Window
Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
Wire Selector
Electrical Ground
In1
Out1
Subsystem
In1
Out1
Subsystem (Configurable)
Probe
Probe

Output the probed signals.

FIG. 3.24 Probe block.

Add a "Probe" block to the schematic (Fig. 3.25).

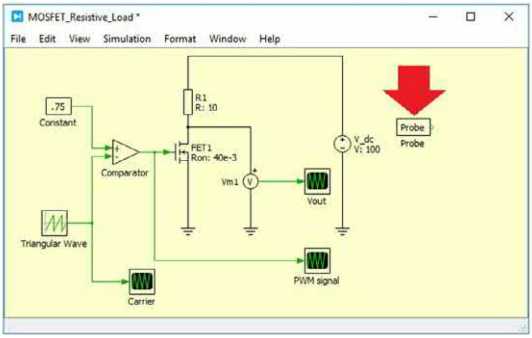
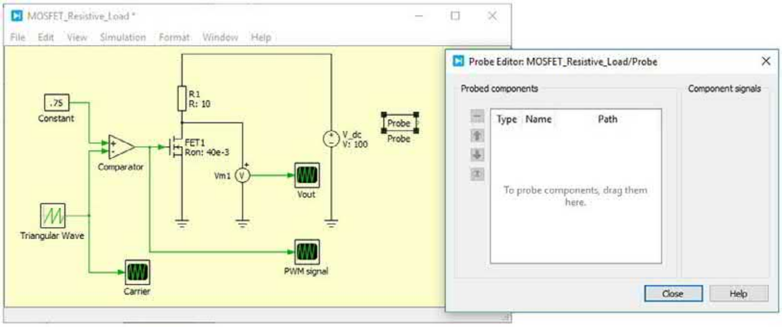

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
.75
Constant
Comparator
R1
R: 10
FET1
Ron: 40e-3
Vm1
V_dc
V: 100
Probe
Probe
Vout
Triangular Wave
PWM signal
Carrier

FIG. 3.25 Adding a probe block to the schematic.

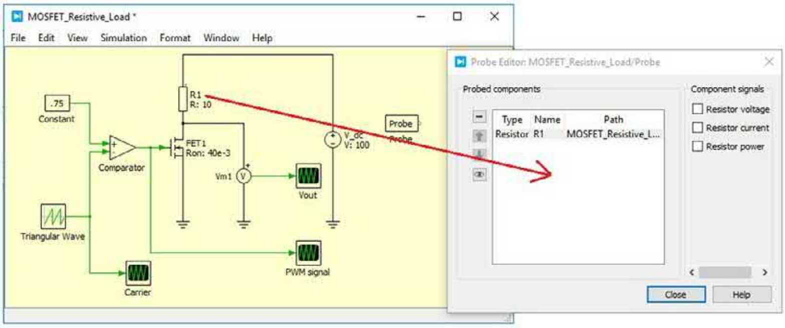
Double click the "Probe" block. The "Probe" block will be opened (Fig. 3.26).


MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
.75
Constant
Comparator
Triangular Wave
Carrier
R1
R: 10
FET1
Ron: 40e-3
Vm1
V
Vout
PWM signal
V_dc
V: 100
Probe
Probe
Probe Editor: MOSFET_Resistive_Load/Probe
Probed components
Component signals
Type Name Path
Close
Help

To probe components, drag them here.

FIG. 3.26 Double clicking on the probe block will open the block.

Drag and drop the resistor R1 into the "Probed components" window (Fig. 3.27).

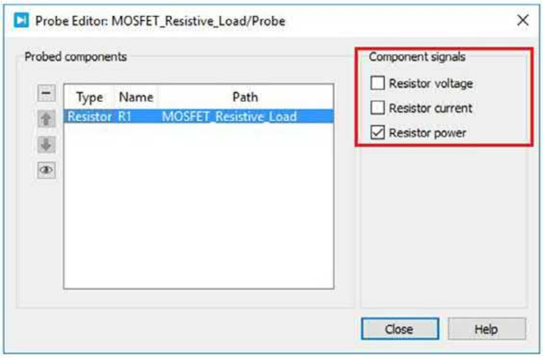

MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
.75
Constant
Comparator
Triangular Wave
Carrier
R1
R: 10
FET1
Ron: 40e-3
Vm1
V
Vout
PWM signal
V_dc
V: 100
Probe
Probe
Probe Editor: MOSFET_Resistive_Load/Probe
Probed components
Component signals
Close
Help

Type	Name	Path
Resistor	R1	MOSFET_Resistive_L...

☐ Resistor voltage
☐ Resistor current
☐ Resistor power

FIG. 3.27 Drag and drop the resistor into the white region.

The "Probe" block will be changed to that shown in Fig. 3.28. Check the "Resistor power" box since we need to measure the dissipated power in the resistor.


Probe Editor: MOSFET_Resistive_Load/Probe
Probed components
Component signals
Close
Help

Type	Name	Path
Resistor	R1	MOSFET_Resistive_Load

☐ Resistor voltage
☐ Resistor current
☑ Resistor power

FIG. 3.28 PLECS shows the measurable quantities for the dropped component.

You can drag and drop other components (passive components, semiconductor switches, electrical machines, etc.) into the "Probe" block and "Probe" block shows you the measurable quantities based on the dropped components. So you can use the "Probe" block with other types of components. It is not limited only to resistors.

Connect the output of the "Probe" block to a scope. We named the scope as "Resistor power" to understand the type of waveform inside it easily (Fig. 3.29).

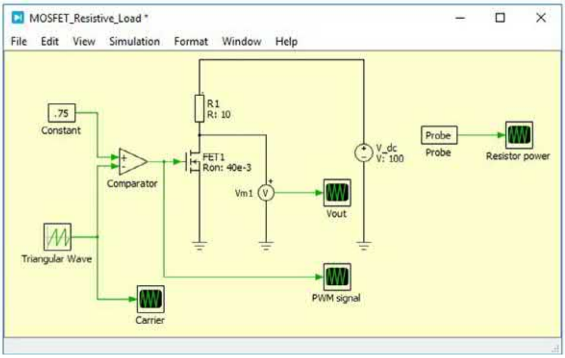

MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
.75
Constant
Comparator
R1
R: 10
FET1
Ron: 40e-3
Vm1
V_dc
V: 100
Vout
Probe
Probe
Resistor power
Triangular Wave
Carrier
PWM signal

FIG. 3.29 Connecting the probe block to a scope.

If we run the simulation, we will see the resistor power waveform (Fig. 3.30).

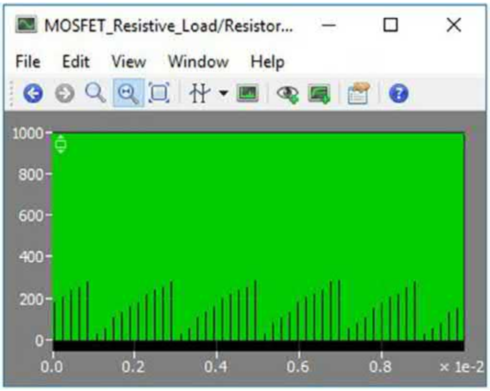

MOSFET_Resistive_Load/Resistor...
File Edit View Window Help
1000
800
600
400
200
0
0.0
0.2
0.4
0.6
0.8
× 1e-2

FIG. 3.30 Resistor power waveform ("Resistor power" scope).

Zoom in the waveform to get a better view using the magnifier icon (Fig. 3.31).

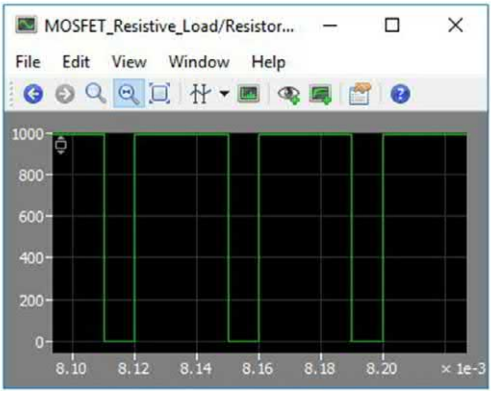
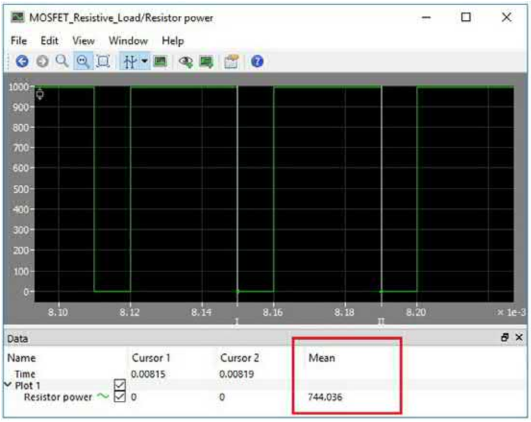

MOSFET_Resistive_Load/Resistor...
File
Edit
View
Window
Help
1000
800
600
400
200
0
8.10
8.12
8.14
8.16
8.18
8.20
× 1e-3

FIG. 3.31 Zooming the resistor power waveform.

You can measure the average value of dissipated power using the aforementioned techniques (Fig. 3.32).


MOSFET_Resistive_Load/Resistor power
File
Edit
View
Window
Help
1000
900
800
700
600
500
400
300
200
100
0
8.10
8.12
8.14
8.16
8.18
8.20
× 1e-3
I
II
Data

Name		Cursor 1	Cursor 2	Mean
Time		0.00815	0.00819	
Plot 1	☑			
Resistor power	☑	0	0	744.036

FIG. 3.32 Average (mean value) of the power waveform.

3.1.4 Subsystem block

Consider the schematic shown in Fig. 3.33. The schematic looks crowded and complicated although it is a simple circuit. We can hide some details and keep the schematics simple with the aid of "Subsystem" block (Fig. 3.34).

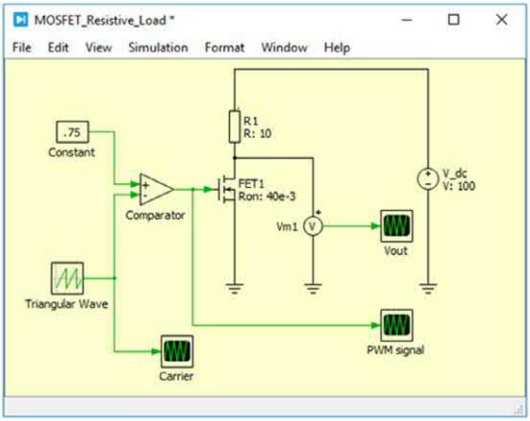
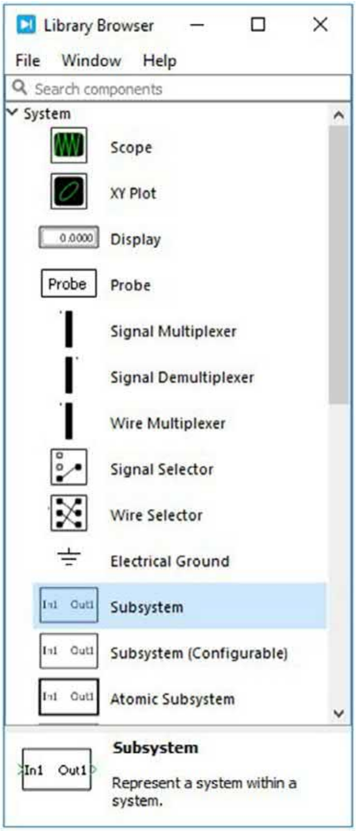

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
.75
Constant
Comparator
Triangular Wave
Carrier
R1
R: 10
FET1
Ron: 40e-3
Vm1
V
Vout
V_dc
V: 100
PWM signal

FIG. 3.33 Schematic seems crowded to some extent.


Library Browser
File
Window
Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
Wire Selector
Electrical Ground
In1 Out1
Subsystem
In1 Out1
Subsystem (Configurable)
In1 Out1
Atomic Subsystem
In1 Out1

Subsystem

Represent a system within a system.

FIG. 3.34 Subsystem block.

In this example, we show how you can make a subsystem for the PWM generation section of the simulation. To do this, left click on empty point in the schematic and draw a rectangle around the PWM generation blocks (Fig. 3.35).

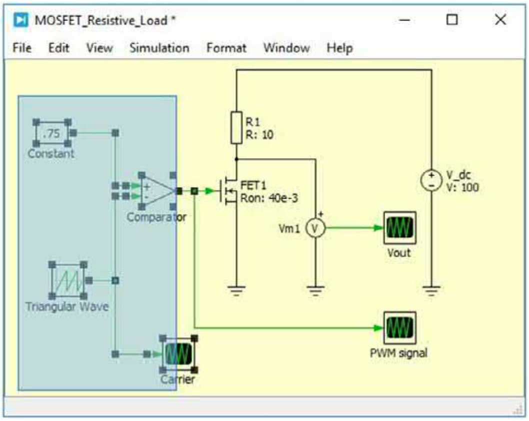

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
.75
Constant
Comparator
Triangular Wave
Carrier
R1
R: 10
FET1
Ron: 40e-3
Vm1
Vout
V_dc
V: 100
PWM signal

FIG. 3.35 Selecting the PWM generation blocks.

After releasing the mouse button, the blue rectangle will disappear and the blocks inside it will be selected (Fig. 3.36).

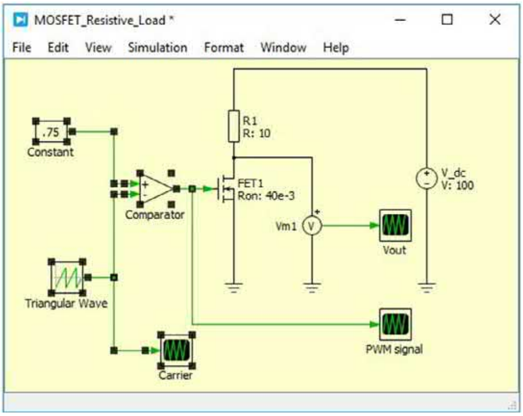

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
.75
Constant
Comparator
Triangular Wave
Carrier
R1
R: 10
FET1
Ron: 40e-3
Vm1
Vout
V_dc
V: 100
PWM signal

FIG. 3.36 Selected blocks.

Cut the selected blocks with the aid of “Cut” (Fig. 3.37).

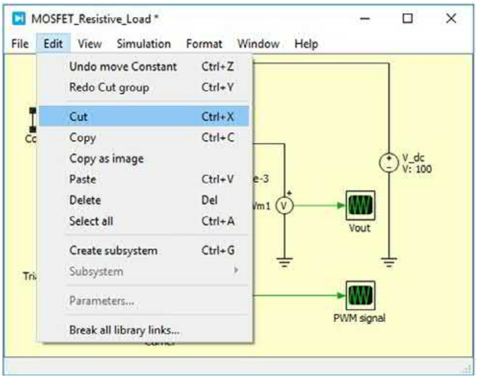

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
Undo move Constant Ctrl+Z
Redo Cut group Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Copy as image
Paste Ctrl+V
Delete Del
Select all Ctrl+A
Create subsystem Ctrl+G
Subsystem
Parameters...
Break all library links...
e-3
Vm1
V_dc
V: 100
Vout
PWM signal

FIG. 3.37 Cutting the selected blocks.

After Clicking the "Cut," selected blocks will be disappeared from the schematic and they are transferred to the Windows clipboard (Fig. 3.38).

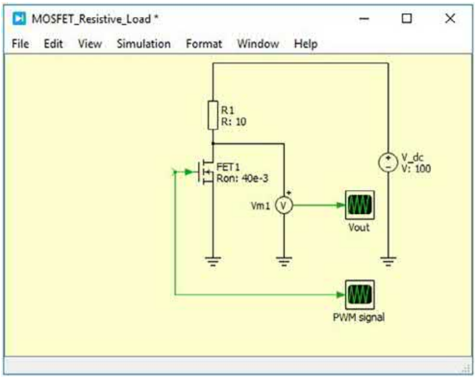

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 10
FET1
Ron: 40e-3
Vm1
V_dc
V: 100
Vout
PWM signal

FIG. 3.38 Schematic after cutting the selected blocks.

Place a subsystem block (see Fig. 3.39) inside the schematic.

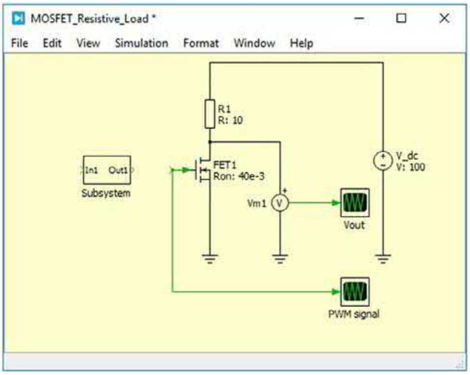

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
In1
Out1
Subsystem
R1
R: 10
FET1
Ron: 40e-3
Vm1
V_dc
V: 100
Vout
PWM signal

FIG. 3.39 Adding a subsystem block to the schematic.

Double click on the placed subsystem label and rename it to “PWM Generator” (Fig. 3.40).

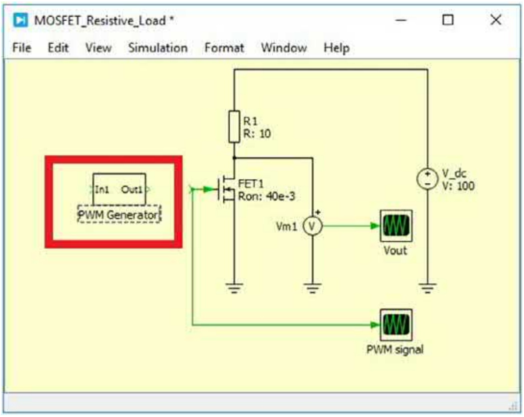

MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
R1
R: 10
In1 Out1
PWM Generator
FET1
Ron: 40e-3
V_dc
V: 100
Vm1
Vout
PWM signal

FIG. 3.40 Renaming the subsystem block.

Double click on the subsystem block to see inside of it. As shown in Fig. 3.41, PLECS by default connects the input and outputs together.

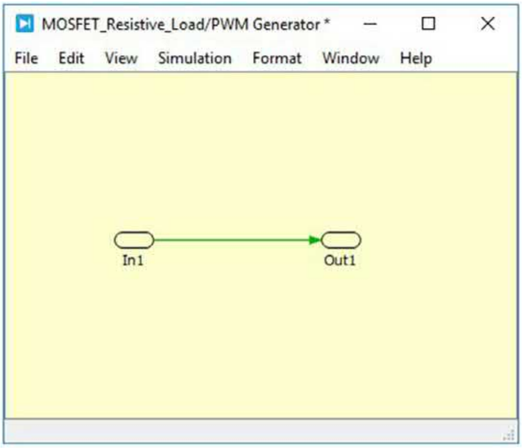

MOSFET_Resistive_Load/PWM Generator *
File Edit View Simulation Format Window Help
In1
Out1

FIG. 3.41 Opening the subsystem block.

Click on an empty point inside the subsystem block and draw a rectangle around the default blocks and press the keyboard Delete key to remove them (Figs. 3.42 and 3.43).

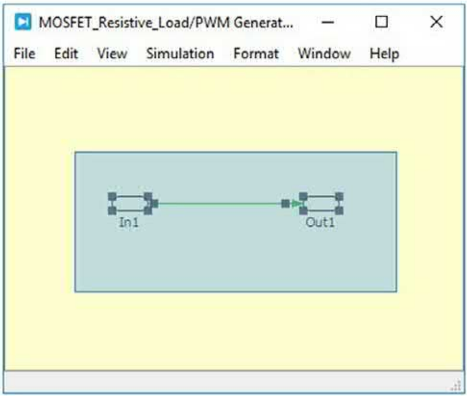

MOSFET_Resistive_Load/PWM Generat...
File Edit View Simulation Format Window Help
In1
Out1

FIG. 3.42 Selecting the content of the subsystem block.

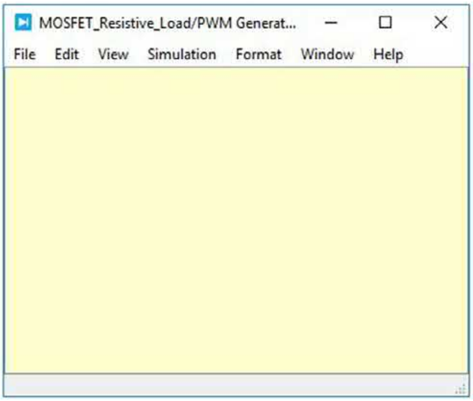

MOSFET_Resistive_Load/PWM Generat...
File Edit View Simulation Format Window Help

FIG. 3.43 Removing the content of the subsystem block.

Paste the content of Windows clipboard (the cut PWM generation circuit) into the subsystem (Figs. 3.44 and 3.45).

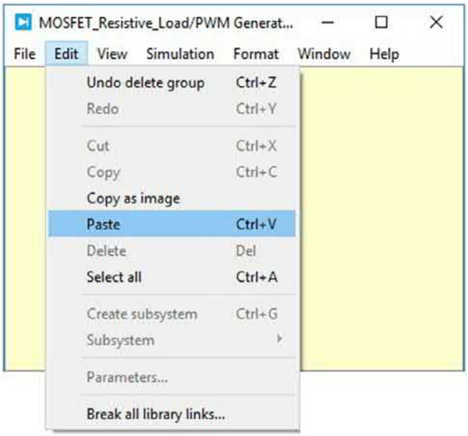

MOSFET_Resistive_Load/PWM Generat...
File Edit View Simulation Format Window Help
Undo delete group Ctrl+Z
Redo Ctrl+Y
Cut Ctrl+X
Copy Ctrl+C
Copy as image
Paste Ctrl+V
Delete Del
Select all Ctrl+A
Create subsystem Ctrl+G
Subsystem
Parameters...
Break all library links...

FIG. 3.44 Paste the PWM generation blocks to the subsystem block.

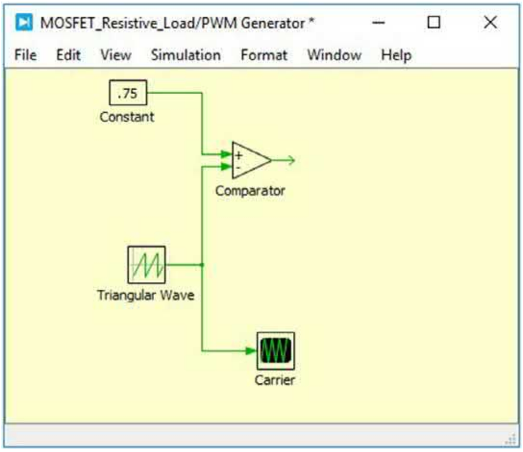

MOSFET_Resistive_Load/PWM Generator *
File Edit View Simulation Format Window Help
.75
Constant
Comparator
Triangular Wave
Carrier

FIG. 3.45 Subsystem block after pasting the PWM generation blocks.

Change the schematic inside the subsystem to that shown in Fig. 3.46. Input/output port can be found under the System section of Library Browser (Fig. 3.47).

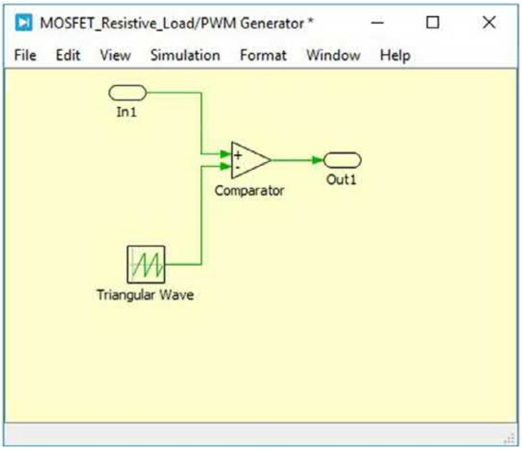

MOSFET_Resistive_Load/PWM Generator *
File Edit View Simulation Format Window Help
In1
Comparator
Out1
Triangular Wave

FIG. 3.46 Adding input and output port to the subsystem.

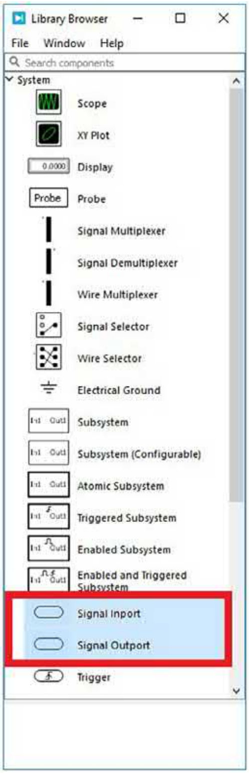

Library Browser
File Window Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
Wire Selector
Electrical Ground
Subsystem
Subsystem (Configurable)
Atomic Subsystem
Triggered Subsystem
Enabled Subsystem
Enabled and Triggered Subsystem
Signal Inport
Signal Outport
Trigger

FIG. 3.47 “Signal Inport” and “Signal Outport” blocks.

Add meaningful labels to the input/output ports(see Fig. 3.48). Close the subsystem window and return to the main schematic. Schematic will appear as shown in Fig. 3.49 now.

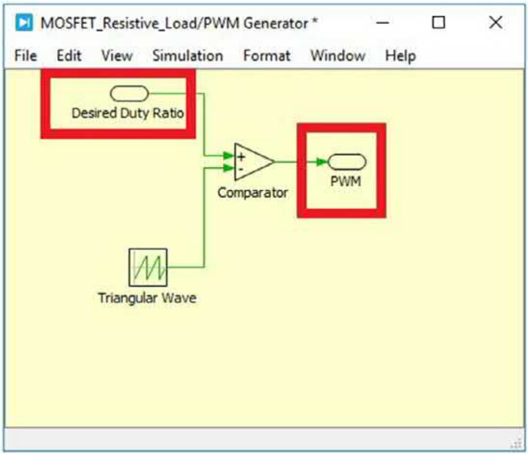

MOSFET_Resistive_Load/PWM Generator *
File Edit View Simulation Format Window Help
Desired Duty Ratio
+
-
Comparator
PWM
Triangular Wave

FIG. 3.48 Renaming the input and output ports.

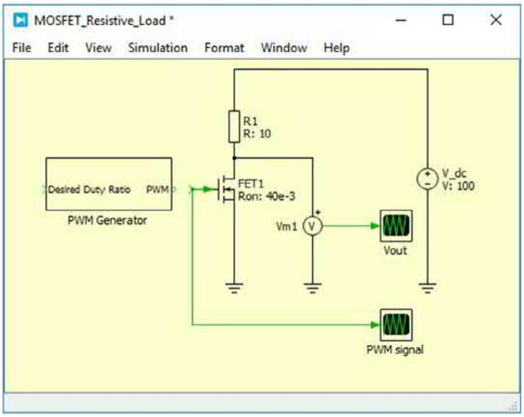

MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
R1
R: 10
Desired Duty Ratio
PWM
PWM Generator
FET1
Ron: 40e-3
V_dc
V: 100
Vm1
V
Vout
PWM signal

FIG. 3.49 Schematic with “PWM Generator” subsystem block.

Change the schematic to that shown in Fig. 3.50.

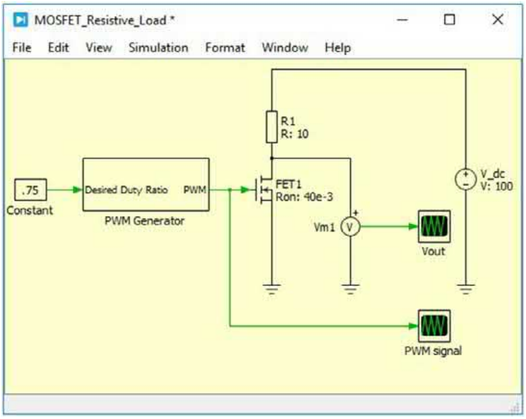

MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
R1
R: 10
.75
Constant
Desired Duty Ratio
PWM
PWM Generator
FET1
Ron: 40e-3
V_dc
V: 100
Vm1
V
Vout
PWM signal

FIG. 3.50 Connecting the “PWM Generator” subsystem to the rest of the circuit.

The schematic shown in Fig. 3.50 is more easily understandable in comparison to Fig. 3.33. Using the subsystem block inside of your simulations is suggested when it is possible. There is no limitation on the number of subsystems blocks inside your simulation.

3.1.5 Measuring the input power

You can measure the input source power in the same way you measured the dissipated power inside the load resistor. To measure the input source power, add a probe block to the schematic (Fig. 3.51).

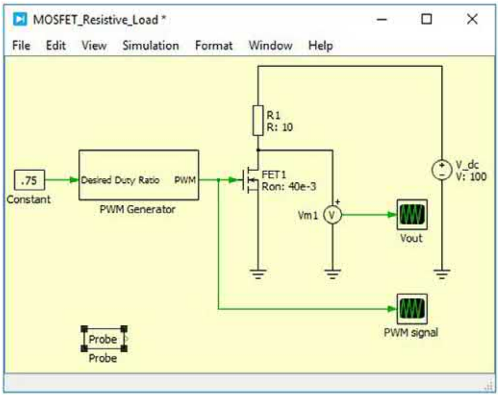
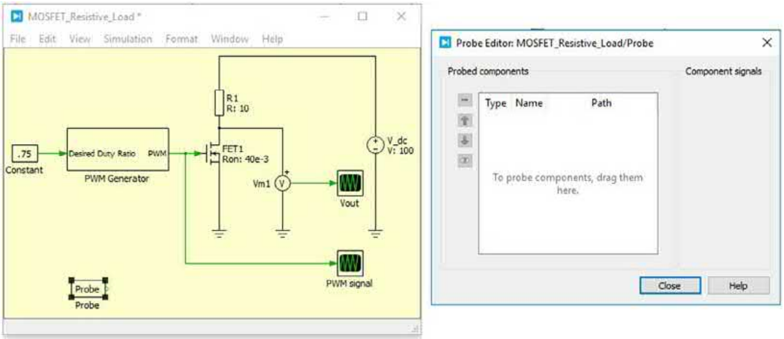

MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
.75
Constant
Desired Duty Ratio PWM
PWM Generator
R1
R: 10
FET1
Ron: 40e-3
Vm1
V
V_dc
V: 100
Vout
PWM signal
Probe
Probe

FIG. 3.51 Adding a probe block to the schematic.

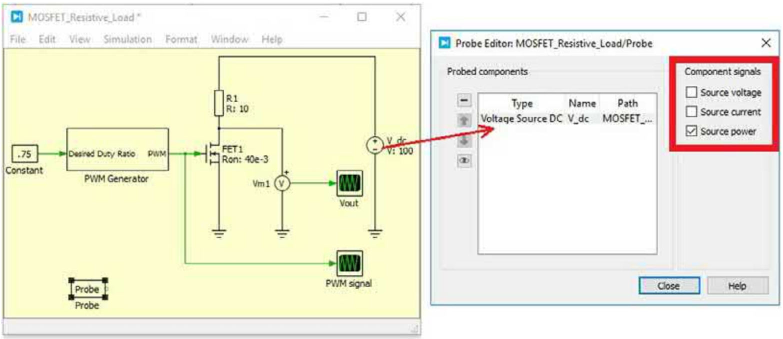
Double click on the probe block to open it (Fig. 3.52).


MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
.75
Constant
Desired Duty Ratio PWM
PWM Generator
R1
R: 10
FET1
Ron: 40e-3
Vm1
V_dc
V: 100
Vout
PWM signal
Probe
Probe
Probe Editor: MOSFET_Resistive_Load/Probe
Probed components
Component signals
Type Name Path
Close
Help

To probe components, drag them here.

FIG. 3.52 Opening the probe block.

Drag and drop the input DC source into the probe block. Check the "Source power" to measure the provided power (Fig. 3.53).


MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
.75
Constant
Desired Duty Ratio PWM
PWM Generator
R1
R: 10
FET1
Ron: 40e-3
Vm1
V_dc
V: 100
Vout
PWM signal
Probe
Probe
Probe Editor: MOSFET_Resistive_Load/Probe
Probed components
Component signals
Close
Help

Type	Name	Path
Voltage Source DC	V_dc	MOSFET_...

☐ Source voltage
☐ Source current
☑ Source power

FIG. 3.53 Drag and drop the input DC source into the white region.

Connect the probe block to a scope block. Name the scope as “Input Power” to remember its content (Fig. 3.54).

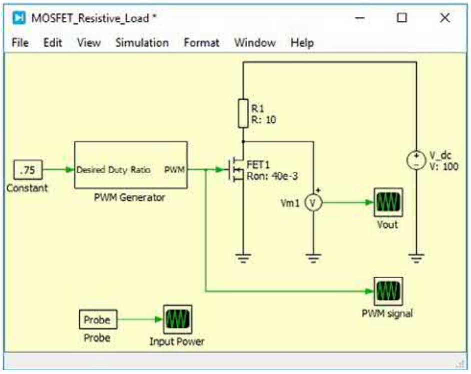

MOSFET_Resistive_Load *
File Edit View Simulation Format Window Help
R1
R: 10
.75
Constant
Desired Duty Ratio PWM
PWM Generator
FET1
Ron: 40e-3
V_dc
V: 100
Vm1
Vout
PWM signal
Probe
Probe
Input Power

FIG. 3.54 Connecting the output of probe block to a scope.

If we run the simulation, we obtain the power waveform shown in Fig. 3.55. Zoom in the waveform to obtain a better view (Fig. 3.56).

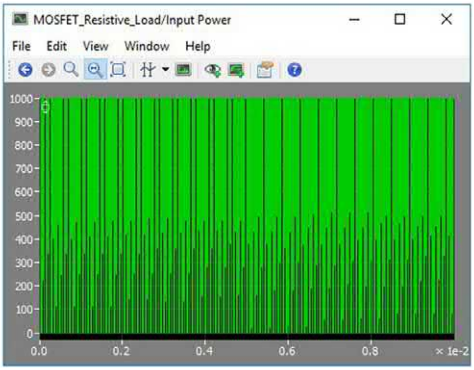

MOSFET_Resistive_Load/Input Power
File Edit View Window Help
1000
900
800
700
600
500
400
300
200
100
0
0.0
0.2
0.4
0.6
0.8
× 1e-2

FIG. 3.55 Input power waveform.

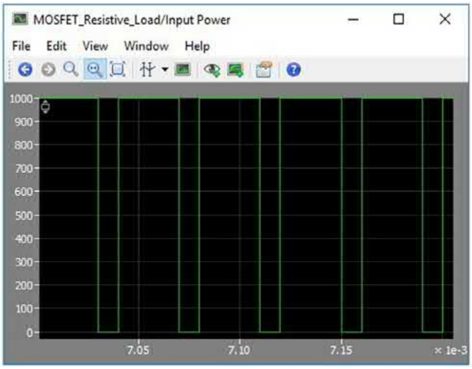
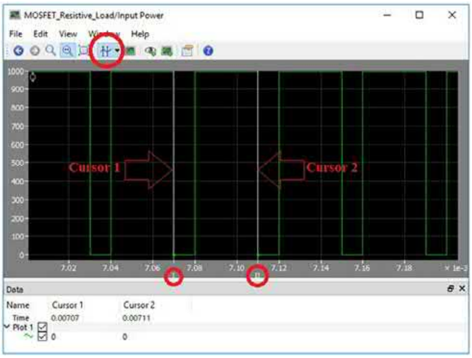

MOSFET_Resistive_Load/Input Power
File Edit View Window Help
1000
900
800
700
600
500
400
300
200
100
0
7.05
7.10
7.15
× 1e-3

FIG. 3.56 Zooming the input power waveform.

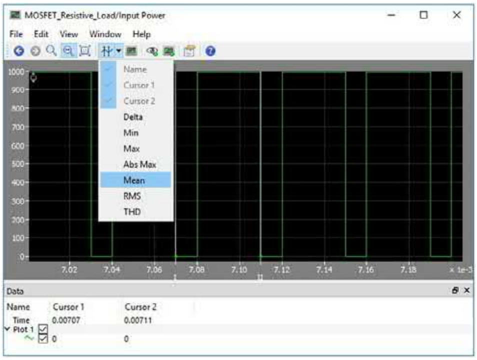
In order to measure the average input power, you must capture a period of input power waveform between the two cursors. Click the cursor icon and capture a period of the input power waveform between the two cursors (Fig. 3.57).


MOSFET_Resistive_Load/Input Power
File
Edit
View
Window
Help
1000
900
800
700
600
500
400
300
200
100
0
Cursor 1
Cursor 2
7.02
7.04
7.06
7.08
7.10
7.12
7.14
7.16
7.18
× 1e-3
Data

Name	Cursor 1	Cursor 2
Time	0.00707	0.00711
Plot 1 ☑		
☑	0	0

FIG. 3.57 Adding cursors to the waveform.

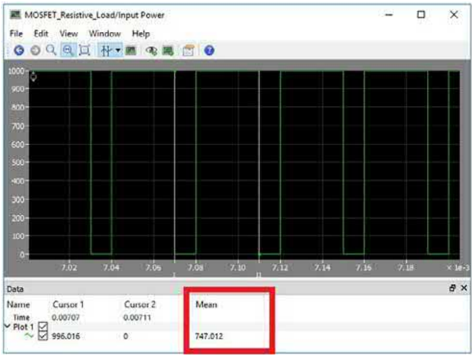
Click the small triangle icon behind the cursor icon and select the "Mean" (Fig. 3.58).


MOSFET_Resistive_Load/Input Power
File
Edit
View
Window
Help
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
RMS
THD
1000
900
800
700
600
500
400
300
200
100
0
7.02
7.04
7.06
7.08
7.10
7.12
7.14
7.16
7.18
× 1e-3
Data

Name	Cursor 1	Cursor 2
Time	0.00707	0.00711
Plot 1 ☑		
☑	0	0

FIG. 3.58 Measuring the average input power.

PLECS calculates the average input power and displays it (Fig. 3.59).


MOSFET_Resistive_Load/Input Power
File
Edit
View
Window
Help
1000
900
800
700
600
500
400
300
200
100
0
7.02
7.04
7.06
7.08
7.10
7.12
7.14
7.16
7.18
× 1e-3
Data

Name	Cursor 1	Cursor 2	Mean
Time	0.00707	0.00711	
Plot 1 ☑			
☑	996.016	0	747.012

FIG. 3.59 Average input power is about 747 W.

The calculated power is a little bit greater than the load resistor power since the MOSFET dissipated some power.

3.1.6 Generating the PWM signal using ready-to-use blocks

Up to now, we simulate the circuit with our own PWM generation circuit. PLECS contains plenty of ready-to-use PWM generator. They can be found in the "Modulator" section of Library Browser (Fig. 3.60).

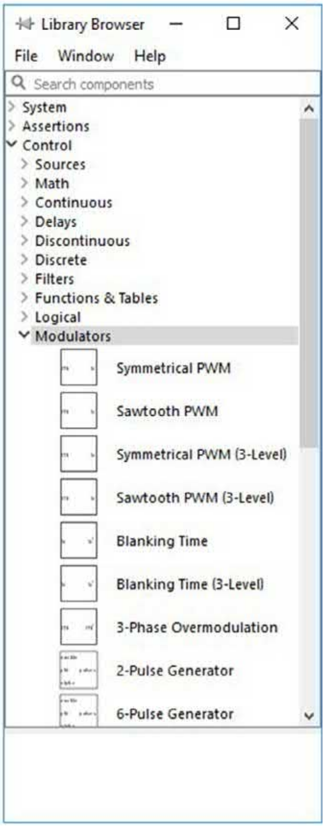
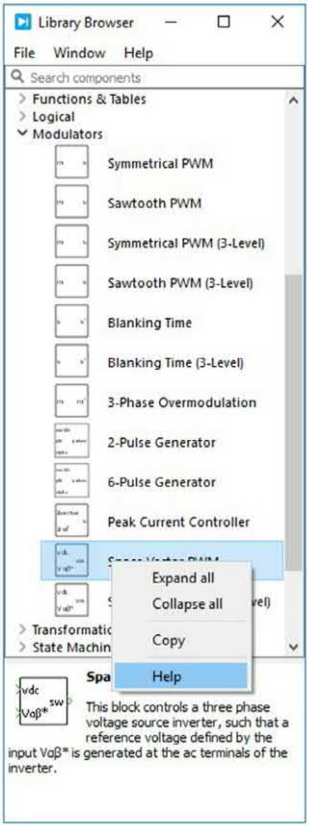

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Symmetrical PWM
Sawtooth PWM
Symmetrical PWM (3-Level)
Sawtooth PWM (3-Level)
Blanking Time
Blanking Time (3-Level)
3-Phase Overmodulation
2-Pulse Generator
6-Pulse Generator

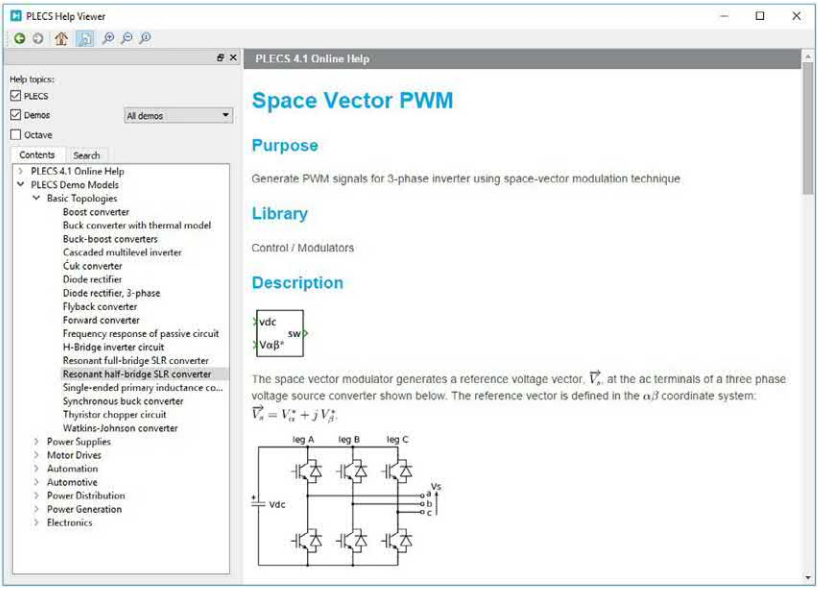
FIG. 3.60 Different modulator blocks are available in PLECS.

You can see the information about each modulator simply by right clicking on the block name and selecting the "Help" from appeared menu (Figs. 3.61 and 3.62).


Library Browser
File
Window
Help
Search components
Functions & Tables
Logical
Modulators
Symmetrical PWM
Sawtooth PWM
Symmetrical PWM (3-Level)
Sawtooth PWM (3-Level)
Blanking Time
Blanking Time (3-Level)
3-Phase Overmodulation
2-Pulse Generator
6-Pulse Generator
Peak Current Controller
Transformatic
State Machin
Expand all
Collapse all
Copy
Help
Spa
vdc
Vαβ*
sw

This block controls a three phase voltage source inverter, such that a reference voltage defined by the input Vαβ* is generated at the ac terminals of the inverter.

FIG. 3.61 You can see the blocks documentation by right click on them and select the Help.


PLECS Help Viewer
Help topics:
☑ PLECS
☑ Demos
All demos
☐ Octave
Contents
Search
PLECS 4.1 Online Help
PLECS Demo Models
Basic Topologies
Boost converter
Buck converter with thermal model
Buck-boost converters
Cascaded multilevel inverter
Ćuk converter
Diode rectifier
Diode rectifier, 3-phase
Flyback converter
Forward converter
Frequency response of passive circuit
H-Bridge inverter circuit
Resonant full-bridge SLR converter
Resonant half-bridge SLR converter
Single-ended primary inductance co...
Synchronous buck converter
Thyristor chopper circuit
Watkins-Johnson converter
Power Supplies
Motor Drives
Automation
Automotive
Power Distribution
Power Generation
Electronics
PLECS 4.1 Online Help
vdc
Vαβ*
sw
leg A
leg B
leg C
Vdc
Vs
a
b
c

Space Vector PWM

Purpose

Generate PWM signals for 3-phase inverter using space-vector modulation technique

Library

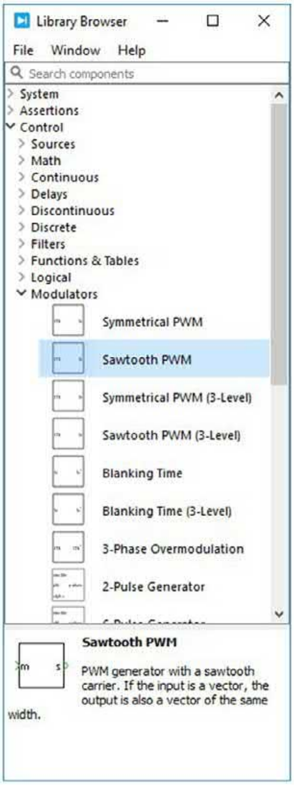
Control / Modulators

Description

The space vector modulator generates a reference voltage vector, $\vec{V}_s$, at the ac terminals of a three phase voltage source converter shown below. The reference vector is defined in the $\alpha\beta$ coordinate system: $\vec{V}_s = V_\alpha^* + j\,V_\beta^*$.

FIG. 3.62 Space Vector PWM modulator block documentation.

Instead of simulating the circuit in the way shown in Fig. 3.50, one can use the “Sawtooth PWM” block as shown in Fig. 3.63 (Fig. 3.64). “Sawtooth PWM” block settings is shown in Fig. 3.65.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Symmetrical PWM
Sawtooth PWM
Symmetrical PWM (3-Level)
Sawtooth PWM (3-Level)
Blanking Time
Blanking Time (3-Level)
3-Phase Overmodulation
2-Pulse Generator
6-Pulse Generator
m
s

Sawtooth PWM

PWM generator with a sawtooth carrier. If the input is a vector, the output is also a vector of the same width.

FIG. 3.63 Sawtooth PWM block.

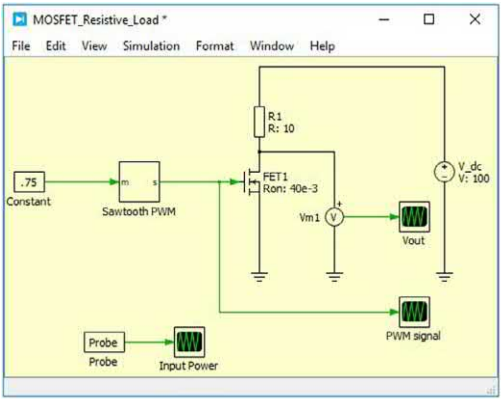
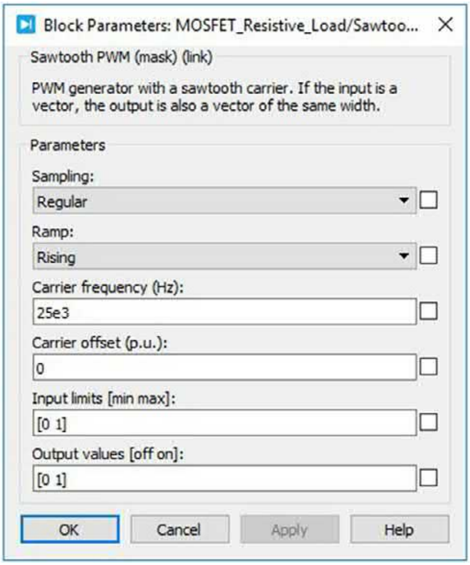

MOSFET_Resistive_Load *
File
Edit
View
Simulation
Format
Window
Help
R1
R: 10
.75
Constant
m
s
Sawtooth PWM
FET1
Ron: 40e-3
V_dc
V: 100
Vm1
V
+
Vout
PWM signal
Probe
Probe
Input Power

FIG. 3.64 Adding the Sawtooth PWM block to the circuit.


Block Parameters: MOSFET_Resistive_Load/Sawtoo...
OK
Cancel
Apply
Help

Sawtooth PWM (mask) (link)

PWM generator with a sawtooth carrier. If the input is a vector, the output is also a vector of the same width.

Parameters

Sampling: Regular ☐

Ramp: Rising ☐

Carrier frequency (Hz): 25e3 ☐

Carrier offset (p.u.): 0 ☐

Input limits [min max]: [0 1] ☐

Output values [off on]: [0 1] ☐

FIG. 3.65 Sawtooth PWM block settings. Using this setting, gate signal has frequency of 25 KHz and duty ratio equal to the "m" input of block. Since "m" is connected to a constant block of 0.75, duty ratio will be 0.75.

3.2 Example 3.2: Uncontrolled single-phase half-wave rectifier

In this example, we will simulate a single-phase half-wave bridge rectifier. Output voltage magnitude is not controllable since the used switch is diode.

3.2.1 Preparing the simulation

Assume a rectifier with RL load as shown in Fig. 3.66. The diode block can be found in the "Power Semiconductors" section of Library Browser (Fig. 3.67). Components settings are shown in Figs. 3.68 and 3.69. Model of a forward biased diode is shown in Fig. 3.70. PLECS assumes open circuit to model reverse biased diodes.

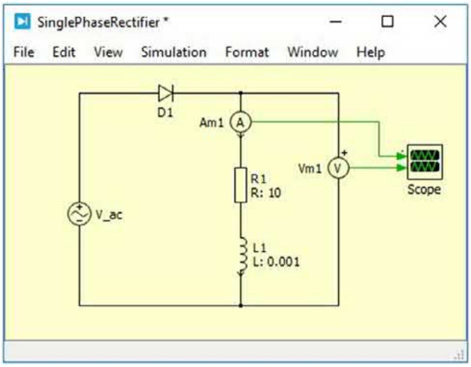
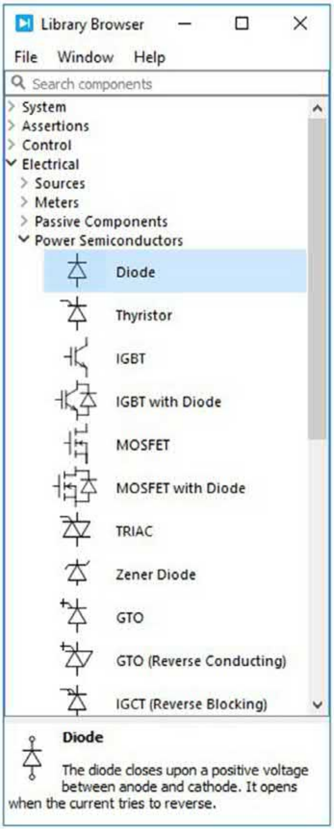

SinglePhaseRectifier *
File
Edit
View
Simulation
Format
Window
Help
D1
Am1
A
Vm1
V
+
R1
R: 10
V_ac
L1
L: 0.001
Scope

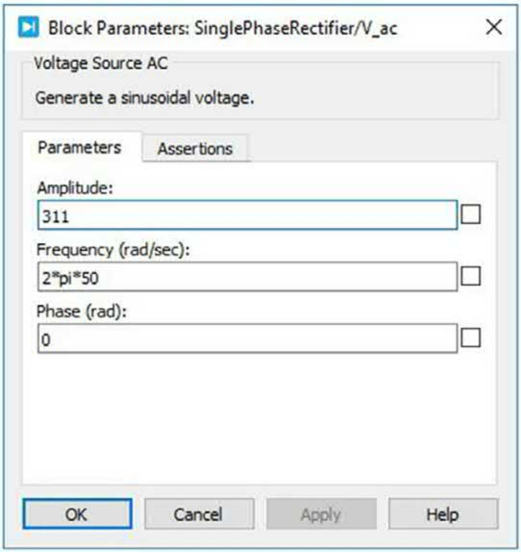
FIG. 3.66 Simple single-phase half-bridge rectifier.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Diode
Thyristor
IGBT
IGBT with Diode
MOSFET
MOSFET with Diode
TRIAC
Zener Diode
GTO
GTO (Reverse Conducting)
IGCT (Reverse Blocking)

Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

FIG. 3.67 Diode block.


Block Parameters: SinglePhaseRectifier/V_ac
Parameters
Assertions
OK
Cancel
Apply
Help

Voltage Source AC

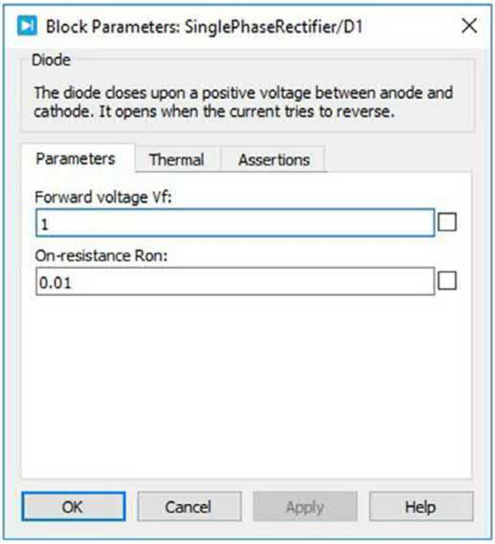
Generate a sinusoidal voltage.

Amplitude: 311 ☐

Frequency (rad/sec): 2*pi*50 ☐

Phase (rad): 0 ☐

FIG. 3.68 Input AC source settings.


Block Parameters: SinglePhaseRectifier/D1
Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

Forward voltage Vf: 1 ☐

On-resistance Ron: 0.01 ☐

FIG. 3.69 Diode settings.

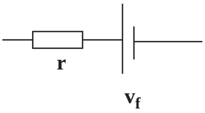
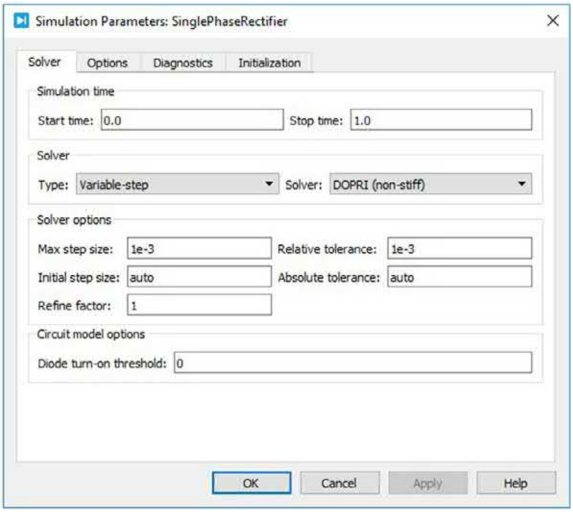


FIG. 3.70 Forward biased diode.

If we do the simulation with the settings shown in Fig. 3.71, we will obtain the results shown in Fig. 3.72.


Simulation Parameters: SinglePhaseRectifier
Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

Simulation time

Start time: 0.0 Stop time: 1.0

Solver

Type: Variable-step Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-3 Relative tolerance: 1e-3

Initial step size: auto Absolute tolerance: auto

Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

FIG. 3.71 Simulation parameters.

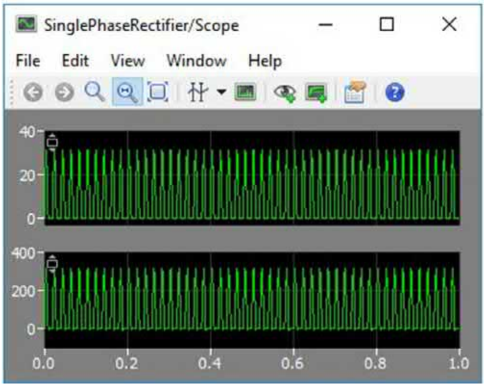

SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
40
20
0
400
200
0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 3.72 Simulation results. Upper axis shows current, while the lower one shows voltage.

Zoom the window to obtain a better view (Fig. 3.73).

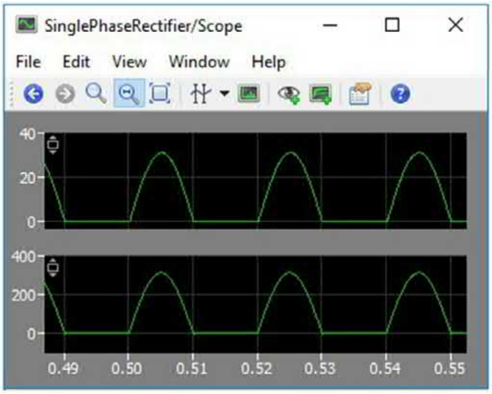
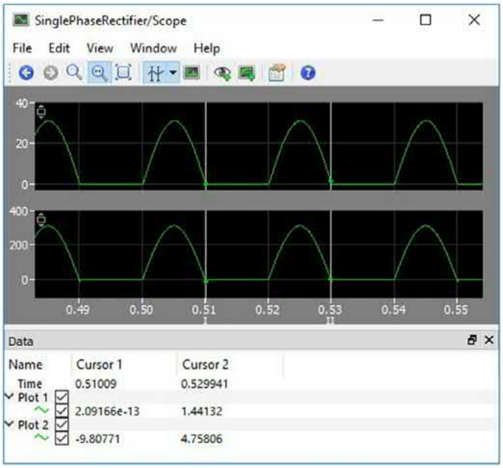

SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
40
20
0
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55

FIG. 3.73 Zooming in the results.

3.2.2 Harmonic content of output

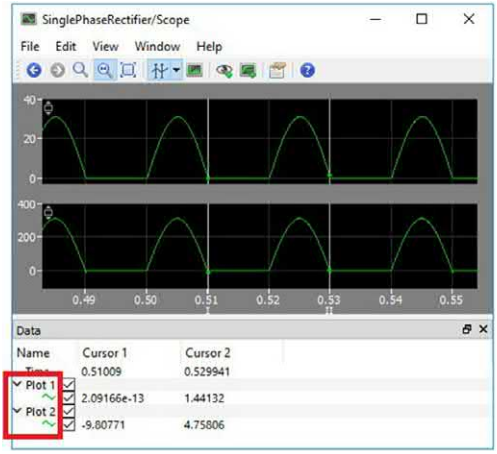
Using PLECS, one can obtain the harmonic content of waveforms easily. For example, in this example, we show how one can obtain the harmonic content of output voltage. Turn on the cursors and capture one period of waveforms. Select the captured waveform from the steady-state part of graph, i.e., where transients finish.

Waveforms inside Fig. 3.74 have no label. So, it is a good idea to show what is what in the first steps. To add labels, double click on the “Plot 1” and “Plot 2” and write suitable names (see Fig 3.75). Upper waveform shows the load current and lower one shows the load voltage (see Fig. 3.76). After writing the suitable names, PLECS shows them on the upper sections of the graph, so one can understand easily what they are.


SinglePhaseRectifier/Scope
File Edit View Window Help
40
20
0
400
200
0
0.49 0.50 0.51 0.52 0.53 0.54 0.55
I
II
Data

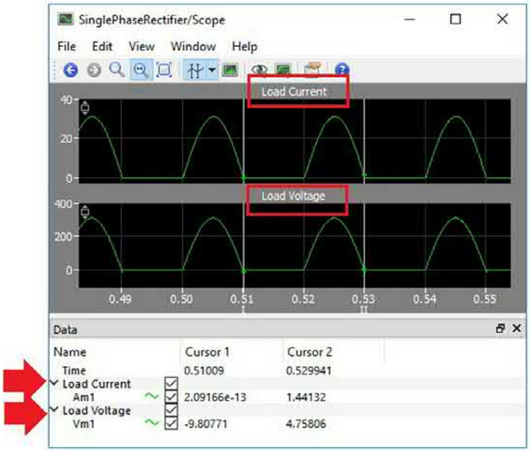
Name		Cursor 1	Cursor 2
Time		0.51009	0.529941
Plot 1	☑		
	☑	2.09166e-13	1.44132
Plot 2	☑		
	☑	-9.80771	4.75806

FIG. 3.74 Capturing one period of waveforms with two cursors.


SinglePhaseRectifier/Scope
File Edit View Window Help
40
20
0
400
200
0
0.49 0.50 0.51 0.52 0.53 0.54 0.55
I
II
Data

Name		Cursor 1	Cursor 2
Time		0.51009	0.529941
Plot 1	☑		
	☑	2.09166e-13	1.44132
Plot 2	☑		
	☑	-9.80771	4.75806

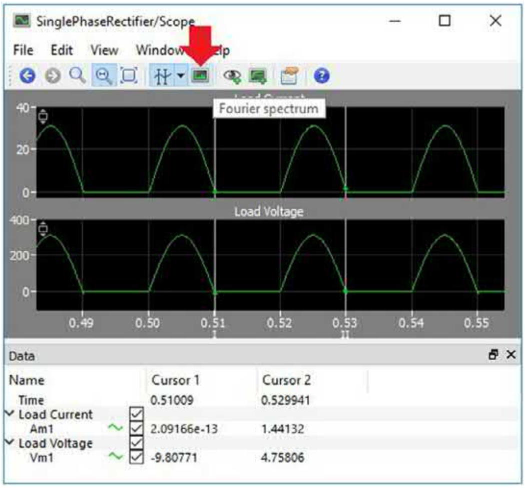
FIG. 3.75 Waveforms seems meaningless since there is no descriptive labels.


SinglePhaseRectifier/Scope
File Edit View Window Help
Load Current
40
20
0
Load Voltage
400
200
0
0.49 0.50 0.51 0.52 0.53 0.54 0.55
I
II
Data

Name		Cursor 1	Cursor 2
Time		0.51009	0.529941
Load Current	☑		
Am1	☑	2.09166e-13	1.44132
Load Voltage	☑		
Vm1	☑	-9.80771	4.75806

FIG. 3.76 Adding descriptive labels to the scope.

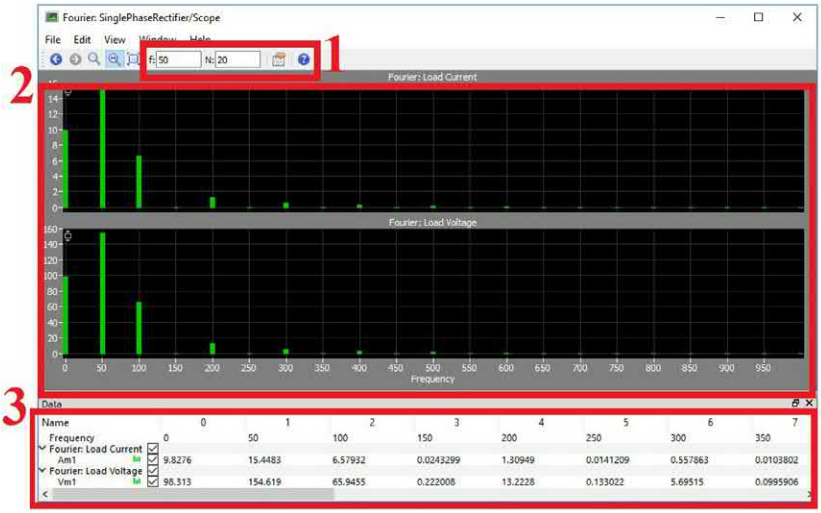
We are ready to do the harmonic analysis now. To do this, click the “Fourier spectrum” icon (Fig. 3.77).


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Fourier spectrum
Load Current
40
20
0
Load Voltage
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

Name		Cursor 1	Cursor 2
Time		0.51009	0.529941
Load Current	☑		
Am1	☑	2.09166e-13	1.44132
Load Voltage	☑		
Vm1	☑	-9.80771	4.75806

FIG. 3.77 Fourier spectrum icon.

PLECS analyses the waveform captured between the cursors and opens a window to show the results. The opened window has three important sections (Fig. 3.78).


Fourier: SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
f: 50
N: 20
1
2
3
Fourier: Load Current
14
12
10
8
6
4
2
0
Fourier: Load Voltage
160
140
120
100
80
60
40
20
0
0
50
100
150
200
250
300
350
400
450
500
550
600
650
700
750
800
850
900
950
Frequency
Data

Name		0	1	2	3	4	5	6	7
Frequency		0	50	100	150	200	250	300	350
Fourier: Load Current	☑								
Am1	☑	9.8276	15.4483	6.57932	0.0243299	1.30949	0.0141209	0.557863	0.0103802
Fourier: Load Voltage	☑								
Vm1	☑	98.313	154.619	65.9455	0.222008	13.2228	0.133022	5.69515	0.0995906

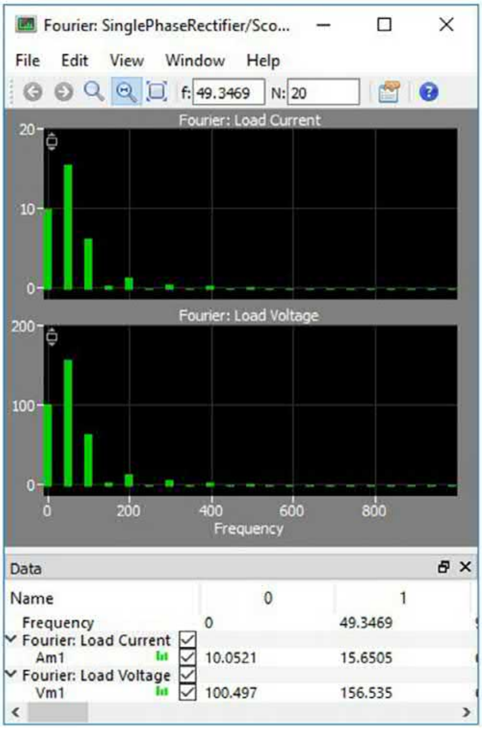
FIG. 3.78 Fourier analysis result.

First section is composed of two text boxes: "f:" and "N:" text boxes. "f:" text box shows the fundamental frequency of waveform captured between the two cursors. "N:" box shows the number of harmonics involved in computation. By default, it is set to 20. Means PLECS calculate the harmonics up to 20th harmonic.

Second section shows the magnitude of harmonic graphically. It does not show the magnitude in dB. For instance, if waveform contains 110 sin $(2 \times \pi \times 1000)$, you will 110 at 1000 Hz not $20 \times \log(110) = 40.83$ dB.

Third section shows the harmonics magnitudes using numbers, so you can read them easily.

Some time you cannot capture exactly one period of output waveform between the two cursors. Assume a situation like that shown in Fig. 3.79. In this case, the fundamental frequency is calculated as 49.3469 Hz. instead of 50 Hz.

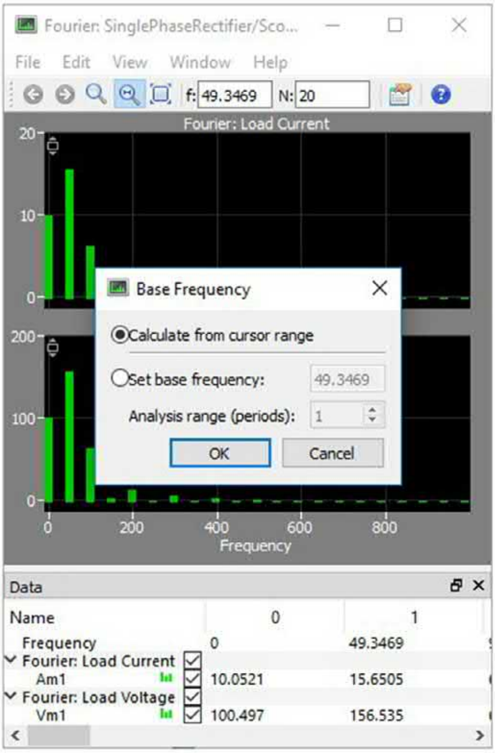

Fourier: SinglePhaseRectifier/Sco...
File
Edit
View
Window
Help
Fourier: Load Current
20
10
0
Fourier: Load Voltage
200
100
0
0
200
400
600
800
Frequency
Data

f: 49.3469 N: 20

Name	0	1
Frequency	0	49.3469
Fourier: Load Current ☑		
Am1 ☑	10.0521	15.6505
Fourier: Load Voltage ☑		
Vm1 ☑	100.497	156.535

FIG. 3.79 "f:" box does not equal 50 Hz.

In such cases, you can click on the "f:" box to correct the fundamental frequency. The window shown in Fig. 3.80 will appear.


Fourier: SinglePhaseRectifier/Sco...
File
Edit
View
Window
Help
Fourier: Load Current
20
10
0
200
100
0
0
200
400
600
800
Frequency
Base Frequency
OK
Cancel
Data

f: 49.3469 N: 20

● Calculate from cursor range

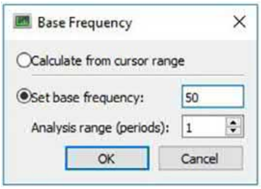
○ Set base frequency: 49.3469

Analysis range (periods): 1

Name		0	1
Frequency		0	49.3469
Fourier: Load Current	☑		
Am1	☑	10.0521	15.6505
Fourier: Load Voltage	☑		
Vm1	☑	100.497	156.535

FIG. 3.80 Base Frequency window.

Select "Set base frequency:" and right click the correct frequency. Click "OK" to apply the changes (Fig. 3.81).


Base Frequency
OK
Cancel

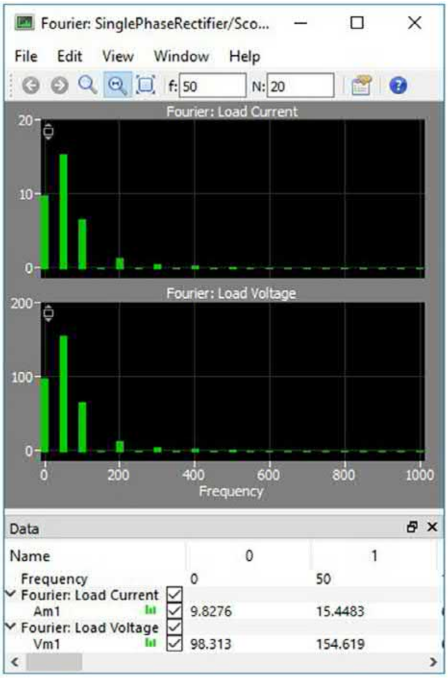
○ Calculate from cursor range

● Set base frequency: 50

Analysis range (periods): 1

FIG. 3.81 Setting the fundamental frequency.

As shown in Fig. 3.82, the fundamental frequency will changes to the new value. You can set the desired number of harmonics to be analyzed by clicking on the "N:" box and write the desired number of harmonics.


Fourier: SingleScope/Sco...
File
Edit
View
Window
Help
Fourier: Load Current
20
10
0
Fourier: Load Voltage
200
100
0
0
200
400
600
800
1000
Frequency
Data

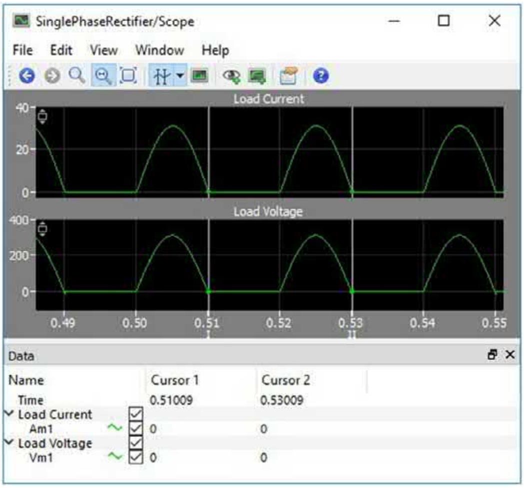
f: 50 N: 20

Name	0	1
Frequency	0	50
Fourier: Load Current ☑		
Am1 ☑	9.8276	15.4483
Fourier: Load Voltage ☑		
Vm1 ☑	98.313	154.619

FIG. 3.82 Analysis results.

3.2.3 Measuring the RMS values of voltages/currents

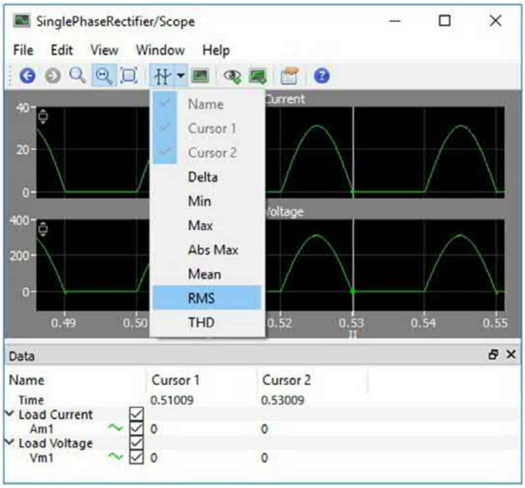
You can measure the RMS value of waveforms easily. Add cursors to the graph and capture one period of waveform (Fig. 3.83).


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Load Current
40
20
0
Load Voltage
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
I
II
Data

Name	Cursor 1	Cursor 2
Time	0.51009	0.53009
Load Current ☑		
Am1 ☑	0	0
Load Voltage ☑		
Vm1 ☑	0	0

FIG. 3.83 Capturing one period of output waveforms.

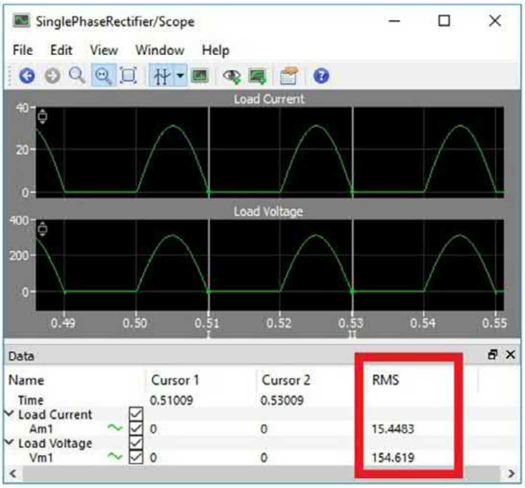
Click the small triangle behind the cursors icon and select the "RMS" (Fig. 3.84).


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
RMS
THD
Current
Voltage
40
20
0
400
200
0
0.49
0.50
0.52
0.53
0.54
0.55
Data

Name		Cursor 1	Cursor 2
Time		0.51009	0.53009
Load Current	☑		
Am1	☑	0	0
Load Voltage	☑		
Vm1	☑	0	0

FIG. 3.84 Calculating the RMS value.

PLECS calculates and shows the RMS value of waveforms. Load current and voltage have RMS of 15.45 A and 154.62 V, respectively (Fig. 3.85).


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Load Current
Load Voltage
40
20
0
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

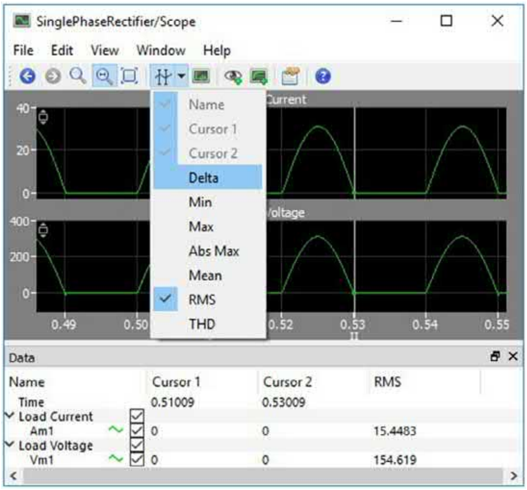
Name		Cursor 1	Cursor 2	RMS
Time		0.51009	0.53009	
Load Current	☑			
Am1	☑	0	0	15.4483
Load Voltage	☑			
Vm1	☑	0	0	154.619

FIG. 3.85 Load current and load voltage RMS values are calculated as 15.4483 A and 154.619 V, respectively.

3.2.4 Capturing a period of output voltage/current

Sometime it is difficult to capture one period of waveforms exactly. In this case, you can ask the PLECS to do that by itself. For example, in the circuit of Fig. 3.66, the load voltage/current waveform period is $\frac{1}{50\,\text{Hz}} = 20\,\text{ms} = 0.02\,\text{s}$, since the source frequency is 50 Hz.

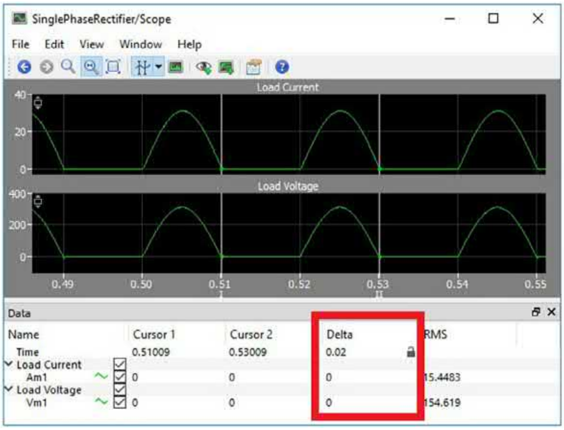
We click on the small triangle behind the cursor icon and select the "Delta" (Fig. 3.86).


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
RMS
THD
Current
Voltage
40
20
0
400
200
0
0.49
0.50
0.52
0.53
0.54
0.55
Data

Name		Cursor 1	Cursor 2	RMS
Time		0.51009	0.53009	
Load Current	☑			
Am1	☑	0	0	15.4483
Load Voltage	☑			
Vm1	☑	0	0	154.619

FIG. 3.86 Measuring the difference between the two cursors.

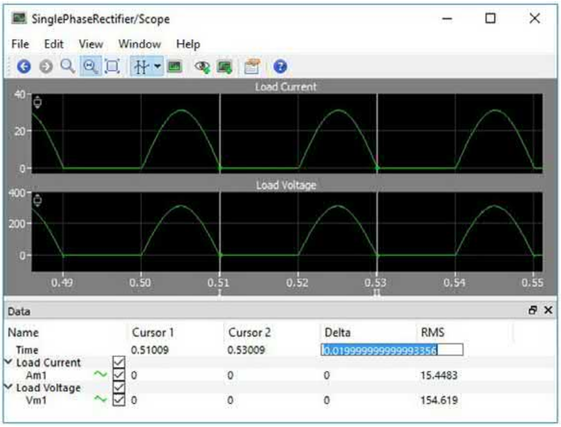
The Delta section is added to the "Data." In this case, the user succeeds to capture exactly 0.02 S of the waveform. Note that the lock icon in the "Delta" section is open. Means you can slide each cursor independently (Fig. 3.87).


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Load Current
Load Voltage
40
20
0
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

Name		Cursor 1	Cursor 2	Delta	RMS
Time		0.51009	0.53009	0.02	
Load Current	☑				
Am1	☑	0	0	0	15.4483
Load Voltage	☑				
Vm1	☑	0	0	0	154.619

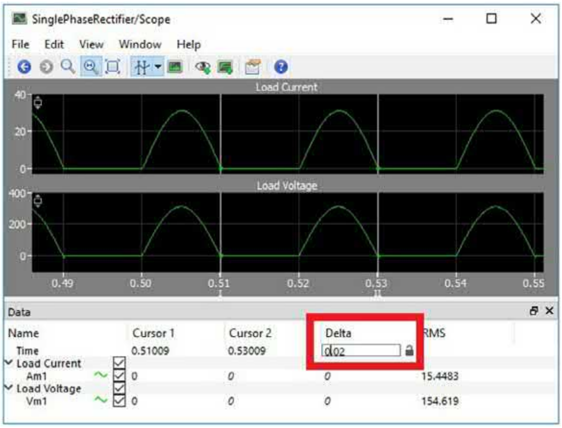
FIG. 3.87 Difference between the two cursors.

If the captured interval is greater or less than the required time length, you can click on the shown value and right the correct value. For example, assume that instead of .02 s you captured .0187 s. in this case you click on the current value and enter the correct value. After writing the correct value, press Enter key on your keyboard (Figs. 3.88 and 3.89).


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Load Current
40
20
0
Load Voltage
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

Name		Cursor 1	Cursor 2	Delta	RMS
Time		0.51009	0.53009	0.0199999999999993356	
Load Current	☑				
Am1	☑	0	0	0	15.4483
Load Voltage	☑				
Vm1	☑	0	0	0	154.619

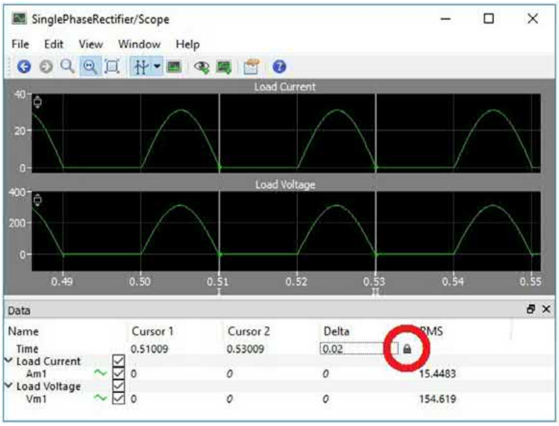
FIG. 3.88 Double click the Delta box to enter the desired difference.


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Load Current
40
20
0
Load Voltage
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

Name		Cursor 1	Cursor 2	Delta	RMS
Time		0.51009	0.53009	0.02	
Load Current	☑				
Am1	☑	0	0	0	15.4483
Load Voltage	☑				
Vm1	☑	0	0	0	154.619

FIG. 3.89 Entering the desired difference.

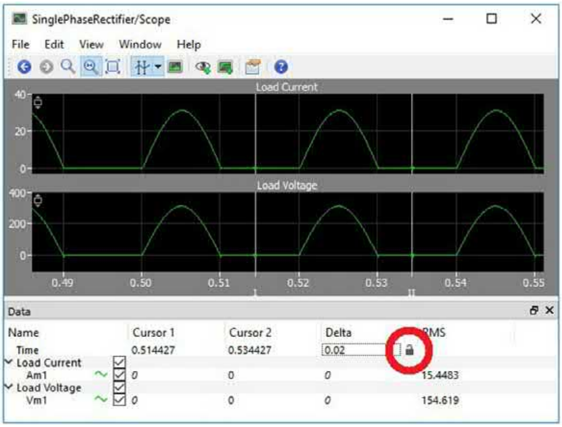
If you double click the lock icon, it will turn into a close lock. It means that the time difference between the two cursors keeps constant. So, if you slide one of the cursors, the other one slides automatically to keep the difference constant at the level written in the text box behind the lock icon (Fig. 3.90).


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Load Current
40
20
0
Load Voltage
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

Name	Cursor 1	Cursor 2	Delta	RMS
Time	0.51009	0.53009	0.02	
Load Current	☑			
Am1	☑ 0	0	0	15.4483
Load Voltage	☑			
Vm1	☑ 0	0	0	154.619

FIG. 3.90 Closed lock shows that the difference between the two cursors keeps constant even if one of the cursor moves.

Double click on the closed lock icon to open the lock (Fig. 3.91).

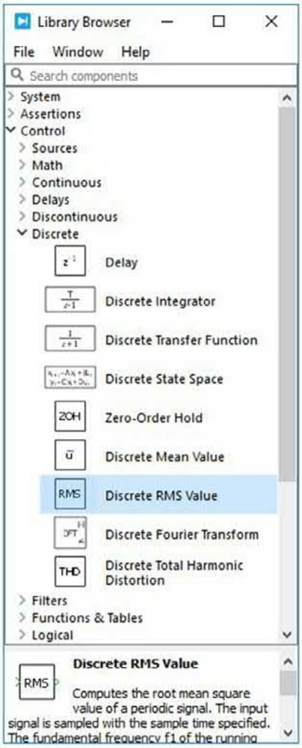

SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Load Current
40
20
0
Load Voltage
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

Name	Cursor 1	Cursor 2	Delta	RMS
Time	0.514427	0.534427	0.02	
Load Current	☑			
Am1	☑ 0	0	0	15.4483
Load Voltage	☑			
Vm1	☑ 0	0	0	154.619

FIG. 3.91 Open lock shows that you can move each of two cursors independent of the other one.

3.2.5 "Discrete RMS value" block

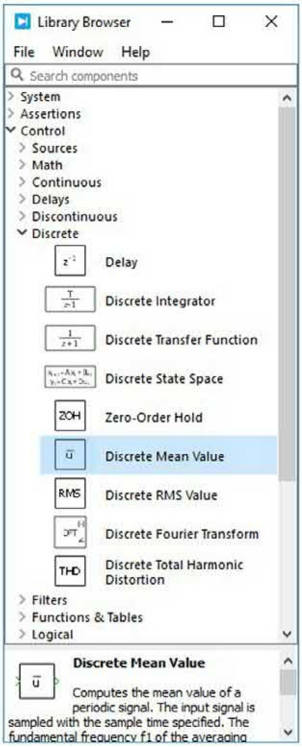
You can measure the RMS and average values of waveforms using the "Discrete RMS value" and "Discrete Mean Value" blocks as well (Figs. 3.92 and 3.93).


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Delay
Discrete Integrator
Discrete Transfer Function
Discrete State Space
Zero-Order Hold
Discrete Mean Value
Discrete RMS Value
Discrete Fourier Transform
Discrete Total Harmonic Distortion
Filters
Functions & Tables
Logical
RMS

Discrete RMS Value

Computes the root mean square value of a periodic signal. The input signal is sampled with the sample time specified. The fundamental frequency f1 of the running

FIG. 3.92 “Discrete RMS Value” block.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Delay
Discrete Integrator
Discrete Transfer Function
Discrete State Space
Zero-Order Hold
Discrete Mean Value
Discrete RMS Value
Discrete Fourier Transform
Discrete Total Harmonic Distortion
Filters
Functions & Tables
Logical

Discrete Mean Value

Computes the mean value of a periodic signal. The input signal is sampled with the sample time specified. The fundamental frequency f1 of the averaging

FIG. 3.93 “Discrete Mean Value” block.

Assume that we want to find the RMS value of load voltage using the "Discrete RMS Value" block. To do this, add a "Discrete RMS Value" block to the schematic (Fig. 3.94).

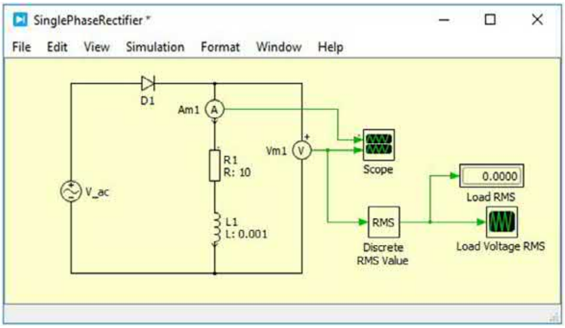
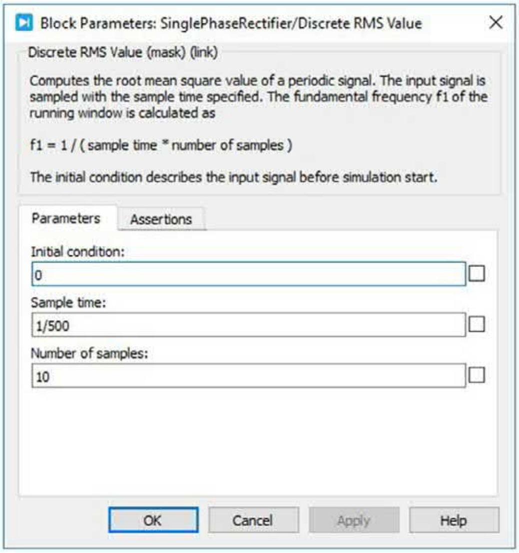

SinglePhaseRectifier *
File
Edit
View
Simulation
Format
Window
Help
D1
Am1
A
Vm1
V
R1
R: 10
L1
L: 0.001
V_ac
Scope
0.0000
Load RMS
RMS
Discrete
RMS Value
Load Voltage RMS

FIG. 3.94 Adding the "Discrete RMS Value" block to the schematic.

Double click on the "Discrete RMS Value" block and do the settings as shown in Fig. 3.95.


Block Parameters: SinglePhaseRectifier/Discrete RMS Value

Discrete RMS Value (mask) (link)

Computes the root mean square value of a periodic signal. The input signal is sampled with the sample time specified. The fundamental frequency f1 of the running window is calculated as

f1 = 1 / (sample time * number of samples)

The initial condition describes the input signal before simulation start.

Parameters | Assertions

Initial condition: 0 ☐

Sample time: 1/500 ☐

Number of samples: 10 ☐

OK
Cancel
Apply
Help

FIG. 3.95 "Discrete RMS Value" block settings.

Since the output voltage frequency is 50 Hz, RMS calculation must be done in a $\frac{1}{50\text{Hz}} = 20\,\text{ms}$ interval. As shown in Fig. 3.95, the fundamental frequency of "Discrete RMS Value" is calculated with the aid of:

$$f = \frac{1}{\text{Sample time} \times \text{Number of samples}}$$

Since we enter the “Sample time:” box as $\frac{1}{500}$ and “Number of samples:” as 10, the fundamental frequency is calculated as:

$$f = \frac{1}{\frac{1}{500} \times 10} = 50\,\text{Hz}$$

So with this setting, PLECS calculates the RMS over the 0.02 s intervals. “Number of samples” can be selected as 10 for most of simulations but you can increase it if you need more accuracy. For example if “Number of samples” is increased to 20, “Sample time:” must be set to 0.001 to keep the fundamental frequency constant at 50 Hz.

If we run the simulation, we obtain the result shown in Fig. 3.96. RMS value is calculated as 154.73 V.

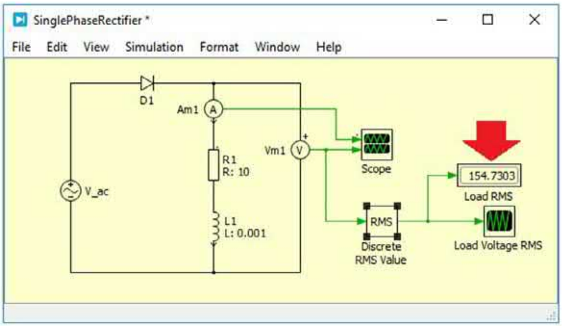

SinglePhaseRectifier *
File Edit View Simulation Format Window Help
D1
Am1
A
R1
R: 10
Vm1
V
V_ac
L1
L: 0.001
Scope
154.7303
Load RMS
RMS
Discrete
RMS Value
Load Voltage RMS

FIG. 3.96 Calculated RMS value for load voltage.

If we open the “Load Voltage RMS” scope, we can read the RMS value as well (Figs. 3.97 and 3.98).

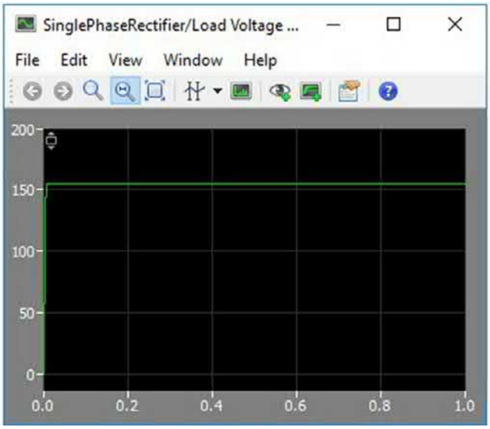
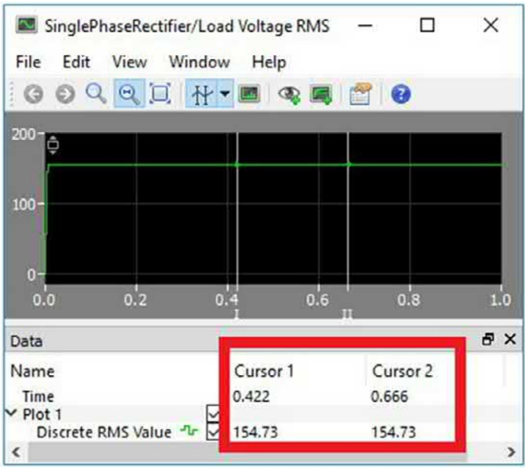

SinglePhaseRectifier/Load Voltage ...
File Edit View Window Help
200
150
100
50
0
0.0
0.2
0.4
0.6
0.8
1.0

FIG. 3.97 “Load Voltage RMS” scope waveform.


SinglePhaseRectifier/Load Voltage RMS
File
Edit
View
Window
Help
200
100
0
0.0
0.2
0.4
0.6
0.8
1.0
I
II
Data

Name	Cursor 1	Cursor 2
Time	0.422	0.666
Plot 1		
Discrete RMS Value ☑	154.73	154.73

FIG. 3.98 Adding cursors to the "Load Voltage RMS" scope to read the RMS value. Transient part of the graph is not important. Put the cursors into the steady-state region of the graph.

3.2.6 "Discrete mean value" block

You can measure the average output by using the "Discerte Mean Value" block as well. To measure the average of output voltage, change the schematic to that shown in Fig. 3.99.

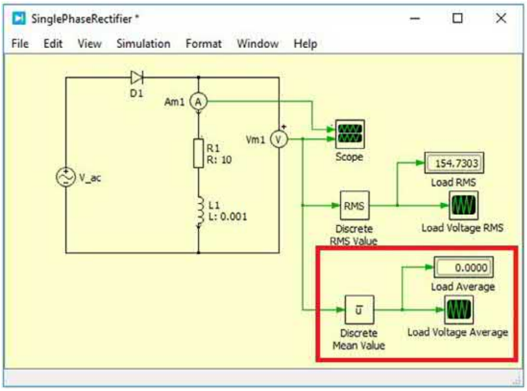
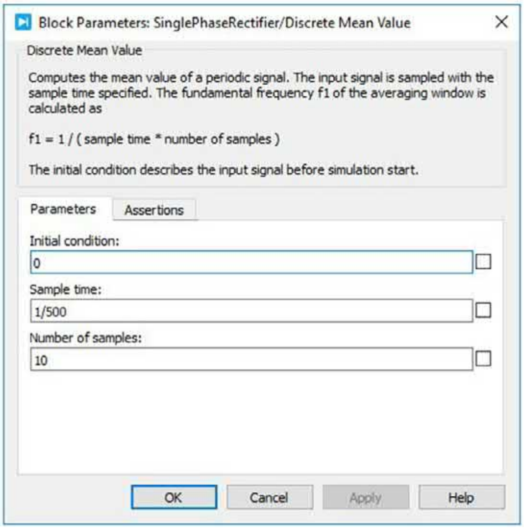

SinglePhaseRectifier *
File
Edit
View
Simulation
Format
Window
Help
D1
Am1
A
Vm1
V
R1
R: 10
V_ac
L1
L: 0.001
Scope
154.7303
Load RMS
RMS
Load Voltage RMS
Discrete
RMS Value
0.0000
Load Average
ū
Discrete
Mean Value
Load Voltage Average

FIG. 3.99 Adding the "Discrete Mean Value" block to the schematic.

Do the "Discrete Mean Value" block settings as shown in Fig. 3.100.



Discrete Mean Value

Computes the mean value of a periodic signal. The input signal is sampled with the sample time specified. The fundamental frequency f1 of the averaging window is calculated as

f1 = 1 / (sample time * number of samples)

The initial condition describes the input signal before simulation start.

Initial condition: 0 ☐

Sample time: 1/500 ☐

Number of samples: 10 ☐

FIG. 3.100 “Discrete Mean Value” block settings.

Since the output voltage frequency is 50 Hz, average (mean value) calculation must be done in a $\frac{1}{50\,\text{Hz}} = 20\,\text{ms}$ interval. As shown in Fig. 3.100, the fundamental frequency of “Discrete Mean Value” is calculated with the aid of:

$$f = \frac{1}{\text{Sample time} \times \text{Number of samples}}$$

Since we enter the “Sample time:” box as $\frac{1}{500}$ and “Number of samples:” as 10, the fundamental frequency is calculated as:

$$f = \frac{1}{\frac{1}{500} \times 10} = 50\,\text{Hz}$$

So with this setting, PLECS calculates the average over the 0.02 s intervals. “Number of samples” can be selected as 10 for most of simulations but you can increase it if you need more accuracy. For example if “Number of samples” is increased to 20, “Sample time:” must be set to 0.001 to keep the fundamental frequency constant at 50 Hz.

If we run the simulation, we obtain the result shown in Fig. 3.101. Mean value is calculated as 95.12 V.

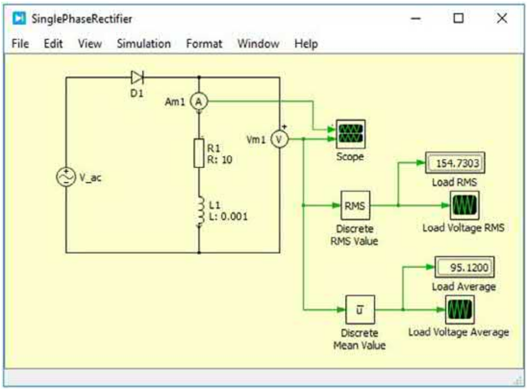
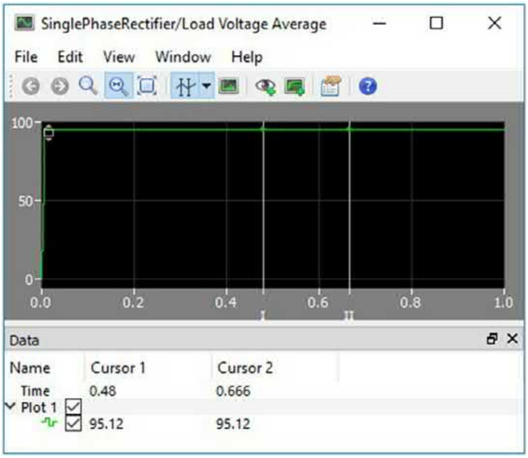

SinglePhaseRectifier
File Edit View Simulation Format Window Help
D1
Am1
A
Vm1
V
R1
R: 10
L1
L: 0.001
V_ac
Scope
154.7303
Load RMS
RMS
Discrete
RMS Value
Load Voltage RMS
95.1200
Load Average
Discrete
Mean Value
Load Voltage Average

FIG. 3.101 Load average voltage is about 95 V.

"Load Voltage Average" scope's waveform is shown in Fig. 3.102.

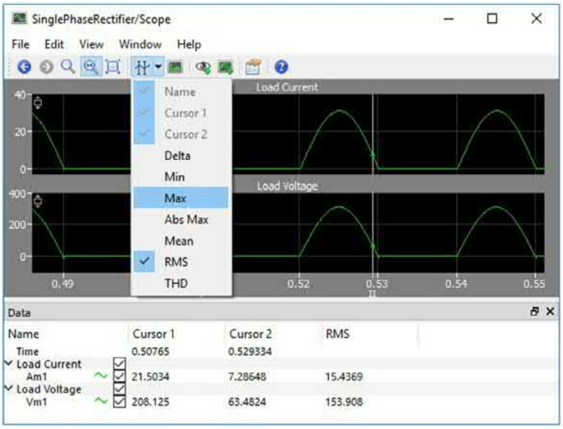

SinglePhaseRectifier/Load Voltage Average
File Edit View Window Help
100
50
0
0.0
0.2
0.4
0.6
0.8
1.0
I
II
Data

Name	Cursor 1	Cursor 2
Time	0.48	0.666
Plot 1 ☑		
☑	95.12	95.12

FIG. 3.102 Adding cursors to the "Load Voltage Average" scope to read the RMS value. Transient part of the graph is not important. Put the cursors into the steady-state region of the graph.

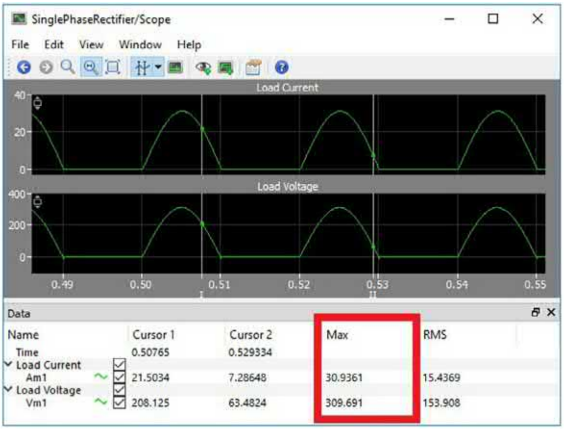
3.2.7 Measuring the maximum/minimum of waveforms shown in the scope block

You can find the maximum/minimum of a waveform easily in PLECS. Capture maximum/minimum between the two cursors and click the Max/Min. PLECS find the maximum/minimum of the interval captured between the two cursors and shows the result (Figs. 3.103–3.106).


SinglePhaseRectifier/Scope
File Edit View Window Help
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
✓ RMS
THD
Load Current
Load Voltage
40
20
0
400
200
0
0.49
0.52
0.53
0.54
0.55
Data

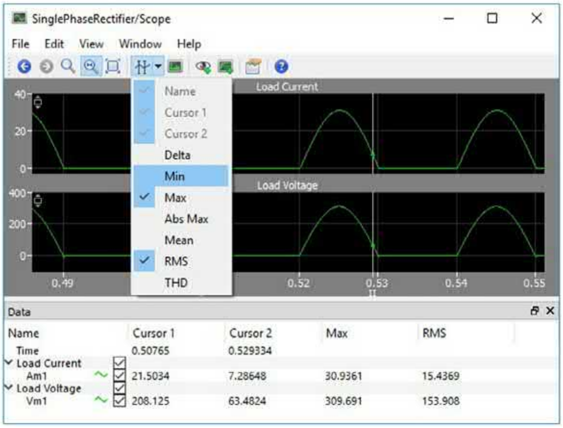
Name	Cursor 1	Cursor 2	RMS
Time	0.50765	0.529334	
Load Current ☑			
Am1 ☑	21.5034	7.28648	15.4369
Load Voltage ☑			
Vm1 ☑	208.125	63.4824	153.908

FIG. 3.103 Calculating the maximum of the waveform captured between the two cursors.


SinglePhaseRectifier/Scope
File Edit View Window Help
Load Current
Load Voltage
40
20
0
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

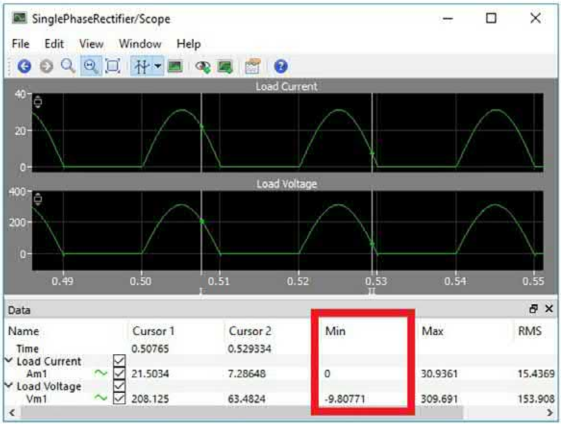
Name	Cursor 1	Cursor 2	Max	RMS
Time	0.50765	0.529334		
Load Current ☑				
Am1 ☑	21.5034	7.28648	30.9361	15.4369
Load Voltage ☑				
Vm1 ☑	208.125	63.4824	309.691	153.908

FIG. 3.104 "Max" column shows the maximum of the waveform captured between the two cursors.


SinglePhaseRectifier/Scope
File Edit View Window Help
Name
Cursor 1
Cursor 2
Delta
Min
✓ Max
Abs Max
Mean
✓ RMS
THD
Load Current
Load Voltage
40
20
0
400
200
0
0.49
0.52
0.53
0.54
0.55
Data

Name	Cursor 1	Cursor 2	Max	RMS
Time	0.50765	0.529334		
Load Current ☑				
Am1 ☑	21.5034	7.28648	30.9361	15.4369
Load Voltage ☑				
Vm1 ☑	208.125	63.4824	309.691	153.908

FIG. 3.105 Calculating the minimum of the waveform captured between the two cursors.


SinglePhaseRectifier/Scope
File
Edit
View
Window
Help
Load Current
40
20
0
Load Voltage
400
200
0
0.49
0.50
0.51
0.52
0.53
0.54
0.55
Data

Name		Cursor 1	Cursor 2	Min	Max	RMS
Time		0.50765	0.529334			
Load Current	☑					
Am1	☑	21.5034	7.28648	0	30.9361	15.4369
Load Voltage	☑					
Vm1	☑	208.125	63.4824	-9.80771	309.691	153.908

FIG. 3.106 "Min" column shows the minimum of the waveform captured between the two cursors.

3.2.8 Obtaining the load instantaneous power

You can find instantaneous load power waveform by multiplying the load current to load voltage. Product block can be found in the Math section of Library Browser (Figs. 3.107 and 3.108).

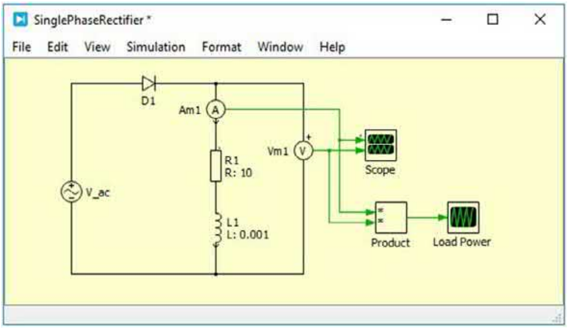
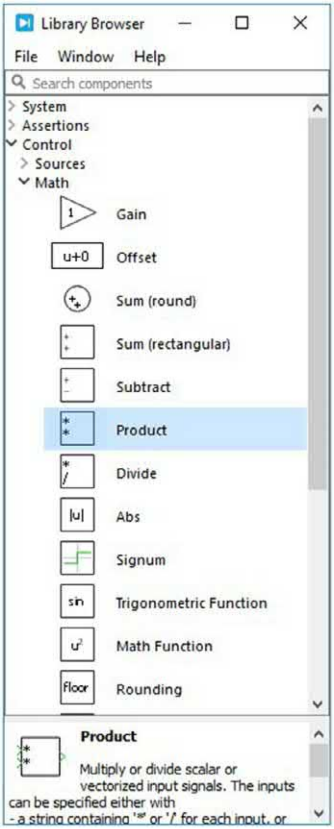

SinglePhaseRectifier *
File
Edit
View
Simulation
Format
Window
Help
D1
Am1
A
Vm1
V
R1
R: 10
V_ac
L1
L: 0.001
Scope
Product
Load Power

FIG. 3.107 Calculating the load power.


Library Browser
File Window Help
Search components
System
Assertions
Control
Sources
Math
1
Gain
u+0
Offset
Sum (round)
Sum (rectangular)
Subtract
Product
Divide
|u|
Abs
Signum
sin
Trigonometric Function
u²
Math Function
floor
Rounding

Product

Multiply or divide scalar or vectorized input signals. The inputs can be specified either with

- a string containing '*' or '/' for each input, or

FIG. 3.108 Product block.

After running the simulation, result shown in Fig. 3.109 will obtained.

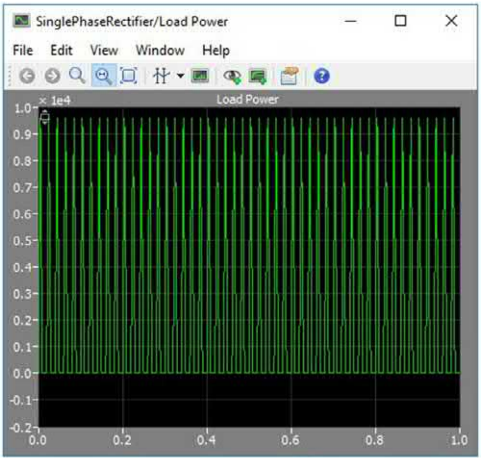


FIG. 3.109 Load power waveform.

Zoom in the graph to obtain a better view (Fig. 3.110).

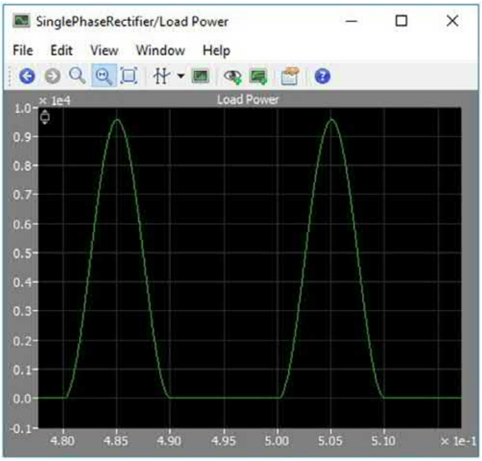
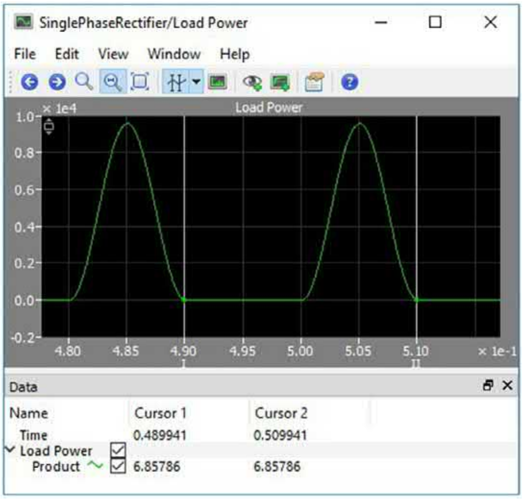

SinglePhaseRectifier/Load Power
File
Edit
View
Window
Help
× 1e4
Load Power
1.0
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0.0
-0.1
4.80
4.85
4.90
4.95
5.00
5.05
5.10
× 1e-1

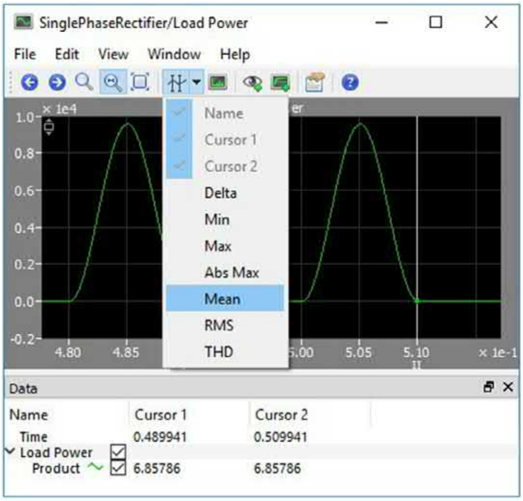
FIG. 3.110 A closer look to load power waveform.

You can calculate the average power by capturing one period of power waveform and clicking the "Mean." PLECS calculates and shows the average load power (Figs. 3.111–3.113).


SinglePhaseRectifier/Load Power
File
Edit
View
Window
Help
× 1e4
Load Power
1.0
0.8
0.6
0.4
0.2
0.0
-0.2
4.80
4.85
4.90
4.95
5.00
5.05
5.10
× 1e-1
I
II
Data

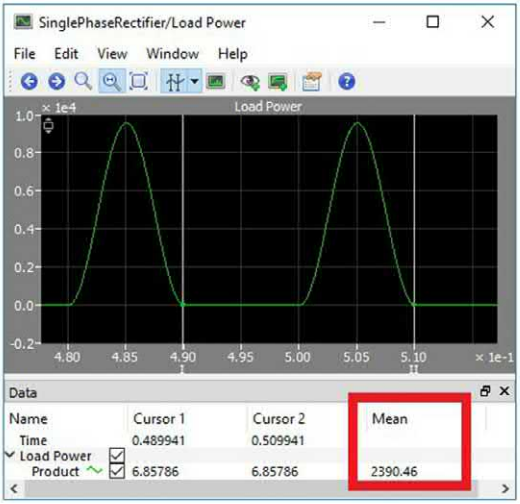
Name	Cursor 1	Cursor 2
Time	0.489941	0.509941
Load Power ☑		
Product ☑	6.85786	6.85786

FIG. 3.111 Capturing a period of the load power.


SinglePhaseRectifier/Load Power
File
Edit
View
Window
Help
× 1e4
1.0
0.8
0.6
0.4
0.2
0.0
-0.2
4.80
4.85
5.00
5.05
5.10
× 1e-1
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
RMS
THD
Data

Name	Cursor 1	Cursor 2
Time	0.489941	0.509941
Load Power ☑		
Product ☑	6.85786	6.85786

FIG. 3.112 Calculating the mean value of load power.


SinglePhaseRectifier/Load Power
File
Edit
View
Window
Help
Load Power
× 1e4
1.0
0.8
0.6
0.4
0.2
0.0
-0.2
4.80
4.85
4.90
4.95
5.00
5.05
5.10
× 1e-1
Data

Name	Cursor 1	Cursor 2	Mean
Time	0.489941	0.509941	
Load Power ☑			
Product ☑	6.85786	6.85786	2390.46

FIG. 3.113 Average power is about 2.39 KW.

3.3 Example 3.3: Single-phase half-wave controlled rectifier

In this example we will simulate a single-phase half-wave rectifier. The output voltages magnitude can be controlled using the thyristor firing angle.

3.3.1 Preparing the simulation

Assume a schematic like Fig. 3.114. The thyristor block can be found in the “Power Semiconductors” section of Library Browser (Fig. 3.115). Input voltage source and pulse generator settings are shown in Figs. 3.116 and 3.117, respectively.

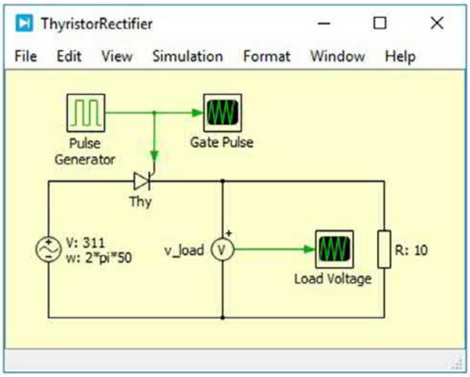
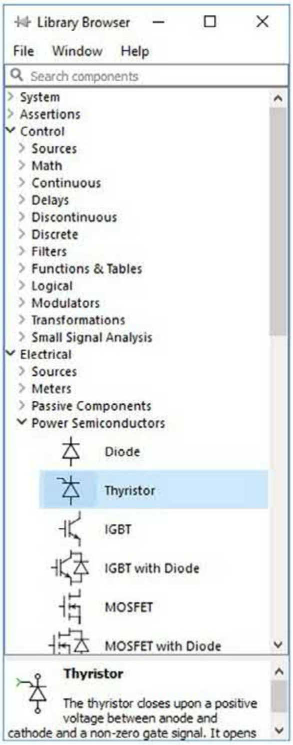

ThyristorRectifier
File Edit View Simulation Format Window Help
Pulse Generator
Gate Pulse
Thy
V: 311
w: 2*pi*50
v_load
V
+
Load Voltage
R: 10

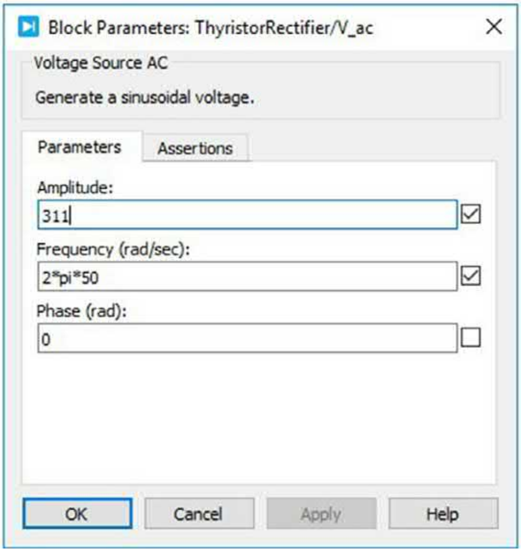
FIG. 3.114 Controlled single-phase half-wave rectifier.


Library Browser
File Window Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Transformations
Small Signal Analysis
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Diode
Thyristor
IGBT
IGBT with Diode
MOSFET
MOSFET with Diode

Thyristor

The thyristor closes upon a positive voltage between anode and cathode and a non-zero gate signal. It opens

FIG. 3.115 Thyristor block.


Block Parameters: ThyristorRectifier/V_ac

Voltage Source AC

Generate a sinusoidal voltage.

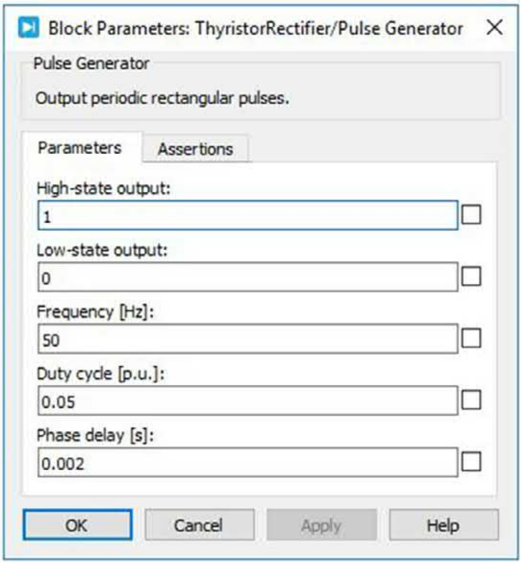
Amplitude: 311 ☑

Frequency (rad/sec): 2*pi*50 ☑

Phase (rad): 0 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.116 Input AC source settings.


Block Parameters: ThyristorRectifier/Pulse Generator

Pulse Generator

Output periodic rectangular pulses.

High-state output: 1 ☐

Low-state output: 0 ☐

Frequency [Hz]: 50 ☐

Duty cycle [p.u.]: 0.05 ☐

Phase delay [s]: 0.002 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.117 Pulse generator settings.

Using the settings shown in Fig. 3.118, the following signal is applied to the thyristor gate.

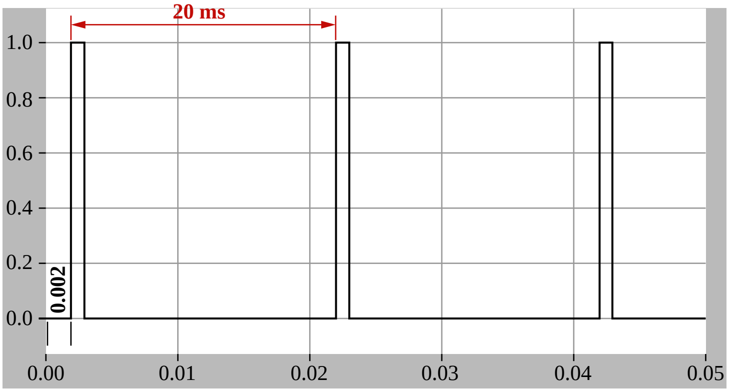
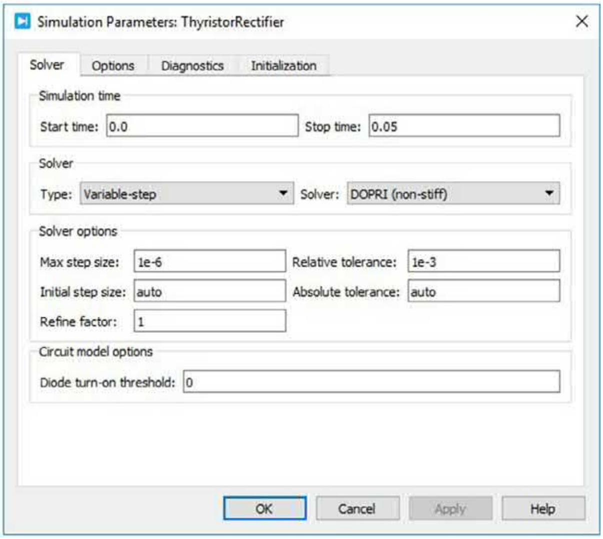


FIG. 3.118 Thyristor gate pulses. Pulse width is $\text{Duty Cycle}\,[\text{p.u.}] \times \frac{1}{\text{Frequency}\,[\text{Hz}]} = 0.05 \times \frac{1}{50} = 1\,\text{ms}$.

If we run the simulation with the settings shown in Fig. 3.119, we obtain the results shown in Figs. 3.120 and 3.121.


Simulation Parameters: ThyristorRectifier
Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

Simulation time

Start time: 0.0
Stop time: 0.05

Solver

Type: Variable-step
Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-6
Relative tolerance: 1e-3
Initial step size: auto
Absolute tolerance: auto
Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

FIG. 3.119 Simulation parameters.

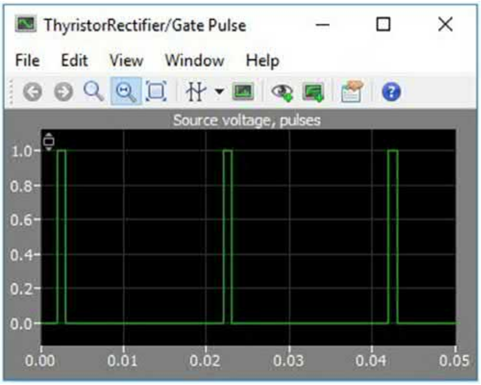

ThyristorRectifier/Gate Pulse
File Edit View Window Help
Source voltage, pulses
1.0
0.8
0.6
0.4
0.2
0.0
0.00
0.01
0.02
0.03
0.04
0.05

FIG. 3.120 Thyristor gate signal.

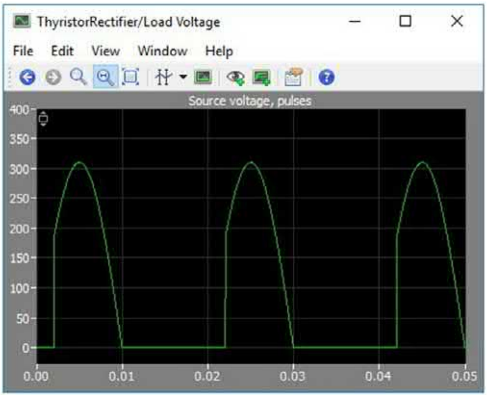

ThyristorRectifier/Load Voltage
File Edit View Window Help
Source voltage, pulses
400
350
300
250
200
150
100
50
0
0.00
0.01
0.02
0.03
0.04
0.05

FIG. 3.121 Load voltage.

3.3.2 Calculating the RMS, mean, max/min, etc.

You can measure the drawn waveform RMS, mean value, max/min, etc. using the techniques studied before.

3.4 Example 3.4: Single-phase full-wave controlled rectifier

In this example, we will simulate a single-phase full-wave rectifier. The output voltages magnitude can be controlled using the thyristor firing angle.

3.4.1 Preparing the simulation

Assume a full-wave controller rectifier shown in Fig. 3.122. The input source settings are shown in Fig. 3.123. The inductor L represents the internal impedance of the source.

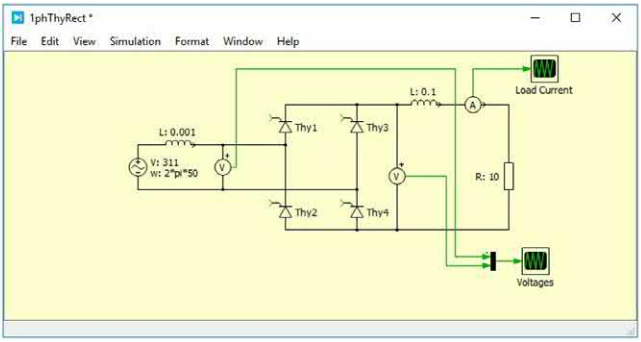
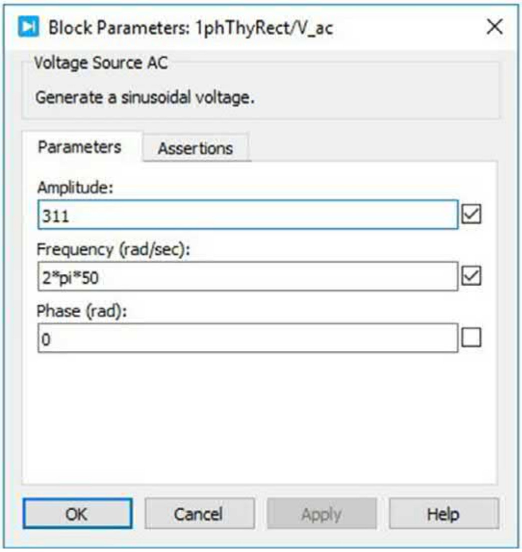

1phThyRect *
File Edit View Simulation Format Window Help
L: 0.001
V: 311
w: 2*pi*50
Thy1
Thy3
Thy2
Thy4
L: 0.1
R: 10
Load Current
Voltages

FIG. 3.122 Controlled single-phase full-wave rectifier.


Block Parameters: 1phThyRect/V_ac

Voltage Source AC

Generate a sinusoidal voltage.

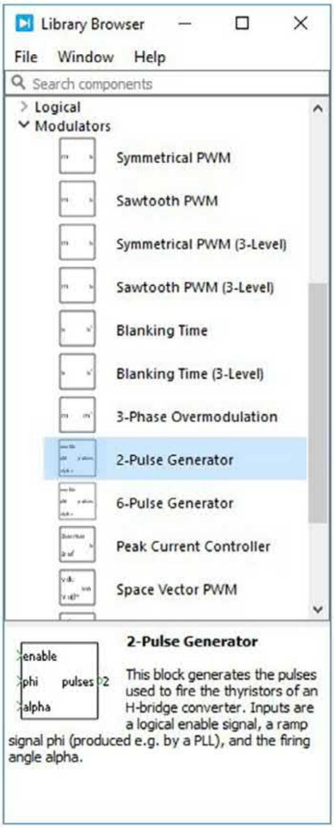
Amplitude: 311 ☑

Frequency (rad/sec): 2*pi*50 ☑

Phase (rad): 0 ☐
Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.123 Input AC source settings.

You can trigger the thyristors using the "2-Pulse Generator" block (Fig. 3.124).


Library Browser
File Window Help
Search components
Logical
Modulators
Symmetrical PWM
Sawtooth PWM
Symmetrical PWM (3-Level)
Sawtooth PWM (3-Level)
Blanking Time
Blanking Time (3-Level)
3-Phase Overmodulation
2-Pulse Generator
6-Pulse Generator
Peak Current Controller
Space Vector PWM
enable
phi
alpha
pulses
2

2-Pulse Generator

This block generates the pulses used to fire the thyristors of an H-bridge converter. Inputs are a logical enable signal, a ramp signal phi (produced e.g. by a PLL), and the firing angle alpha.

FIG. 3.124 2-Pulse Generator block.

Change the schematic to that shown in Fig. 3.125. "Clock" block can be found in the "Sources" section of the Library Browser as shown in Fig. 3.126. New blocks settings are shown in Figs. 3.127–3.129. The desired firing angle is entered into the constant block named "alpha." Here we assumed thyristors are fired at $\frac{\pi}{3}Rad = 60°$.

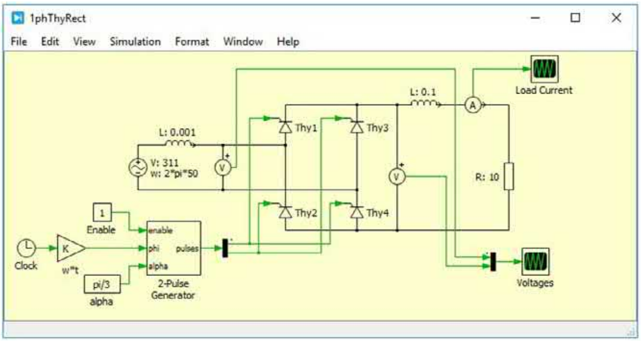
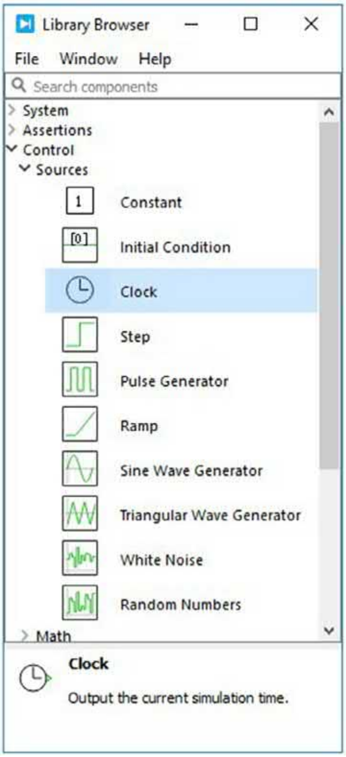

1phThyRect
File Edit View Simulation Format Window Help
L: 0.001
V: 311
w: 2*pi*50
V
Thy1
Thy3
Thy2
Thy4
L: 0.1
A
Load Current
V
R: 10
1
Enable
enable
phi
alpha
pulses
2-Pulse Generator
Clock
K
w*t
pi/3
alpha
Voltages

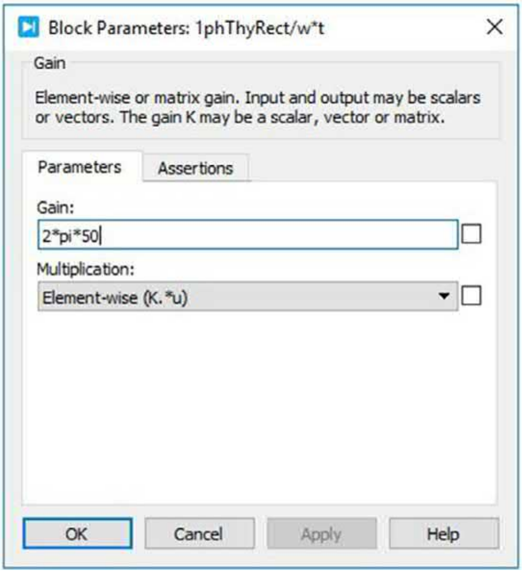
FIG. 3.125 Adding the 2-Pulse Generator block to the schematic.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Constant
Initial Condition
Clock
Step
Pulse Generator
Ramp
Sine Wave Generator
Triangular Wave Generator
White Noise
Random Numbers
Math

Clock

Output the current simulation time.

FIG. 3.126 Clock block.


Block Parameters: 1phThyRect/w*t
Parameters
Assertions
OK
Cancel
Apply
Help

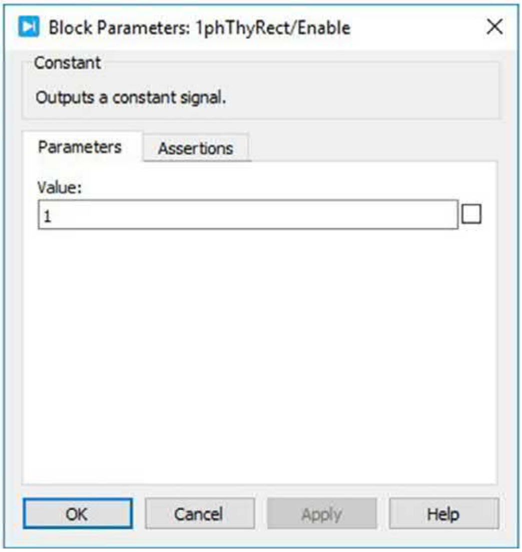
Gain

Element-wise or matrix gain. Input and output may be scalars or vectors. The gain K may be a scalar, vector or matrix.

Gain: 2*pi*50 ☐

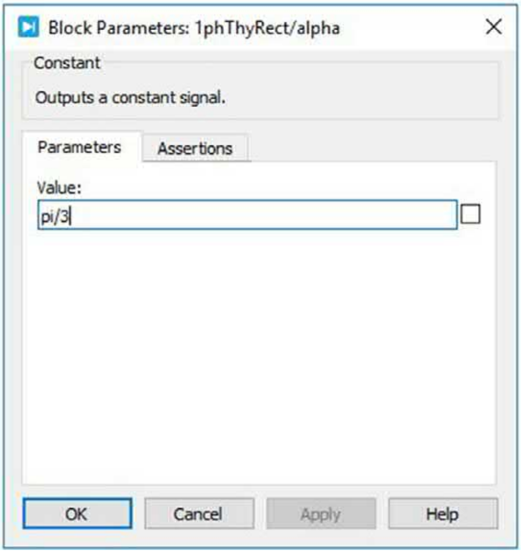
Multiplication: Element-wise (K.*u) ☐

FIG. 3.127 "w*t" block settings (see Fig. 3.125).


Block Parameters: 1phThyRect/Enable
Constant
Outputs a constant signal.
Parameters
Assertions
OK
Cancel
Apply
Help

Value: 1

FIG. 3.128 “Enable” block settings (see Fig. 3.125).


Block Parameters: 1phThyRect/alpha
Constant
Outputs a constant signal.
Parameters
Assertions
OK
Cancel
Apply
Help

Value: pi/3

FIG. 3.129 “Alpha” block settings (see Fig. 3.125).

If we run the simulation, we obtain the results shown in Figs. 3.130 and 3.131.

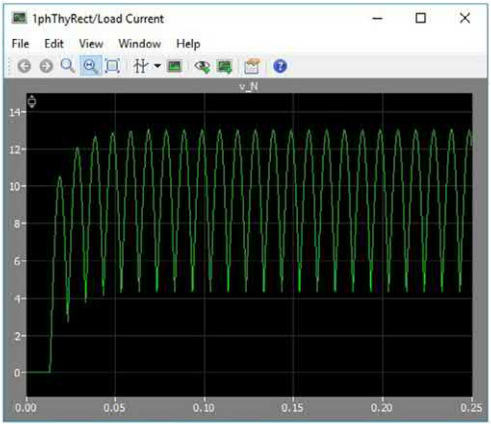

1phThyRect/Load Current
File
Edit
View
Window
Help
v_N
14
12
10
8
6
4
2
0
0.00
0.05
0.10
0.15
0.20
0.25

FIG. 3.130 Load current.

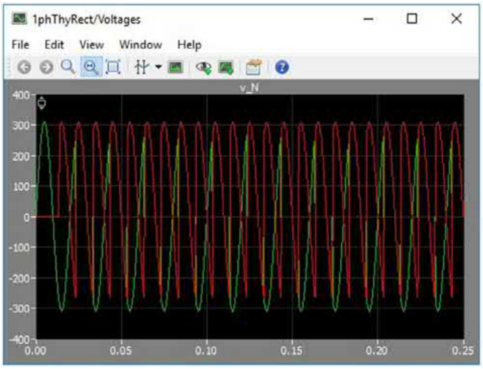

1phThyRect/Voltages
File
Edit
View
Window
Help
v_N
400
300
200
100
0
-100
-200
-300
-400
0.00
0.05
0.10
0.15
0.20
0.25

FIG. 3.131 Input and load voltages.

Zooming in the graph shown in "Voltages" scope to give a better view (Fig. 3.132).

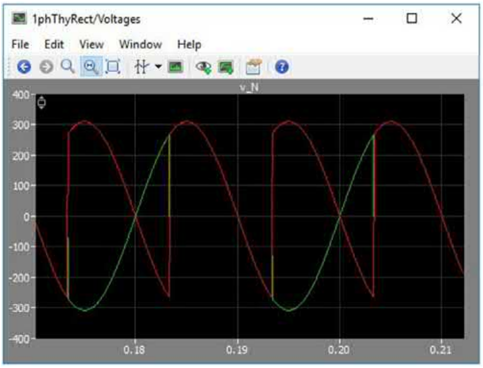

1phThyRect/Voltages
File
Edit
View
Window
Help
v_N
400
300
200
100
0
-100
-200
-300
-400
0.18
0.19
0.20
0.21

FIG. 3.132 Zooming in the waveform.

You can measure the drawn waveform RMS, mean value, max/min, etc. using the techniques studied before.

In the schematic shown in Fig. 3.125, we assumed that input source "Phase (rad):" is 0. If the input source "Phase (rad):" is not zero, the simulation diagram must change to that shown in Fig. 3.133.

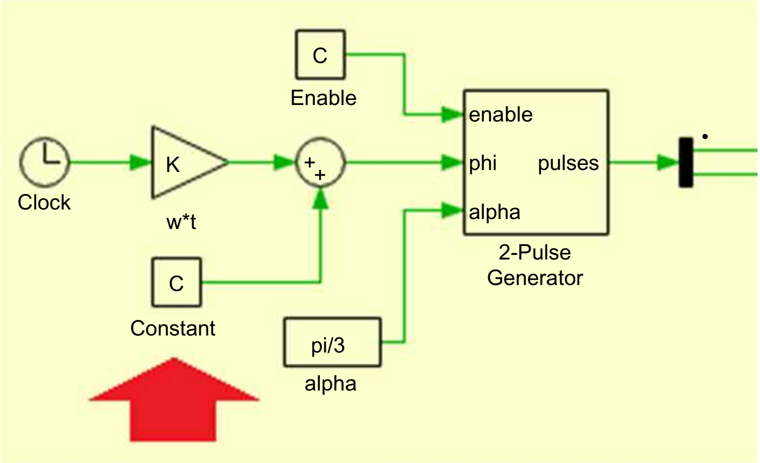
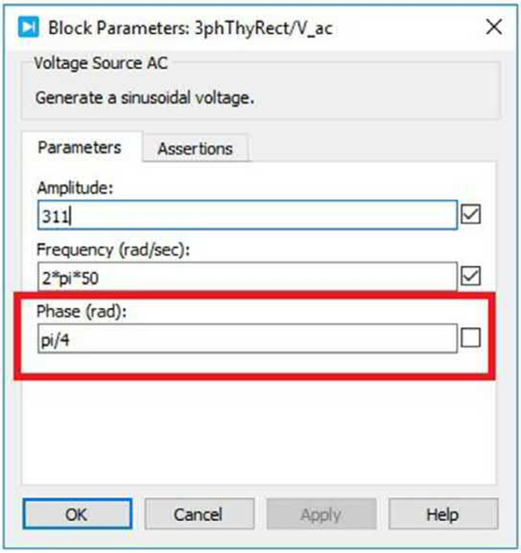

C
Enable
enable
Clock
K
w*t
+
+
phi
pulses
alpha
2-Pulse
Generator
C
Constant
pi/3
alpha

FIG. 3.133 Change in the simulation for Phase (rad) $\neq 0$ case.

For example if we assume that input source "Phase (rad):" is $\frac{\pi}{4}$, then "Constant" block (Fig. 3.134) (see Fig. 3.133) must be filled as shown in Fig. 3.135.


Block Parameters: 3phThyRect/V_ac
×
Parameters
Assertions
OK
Cancel
Apply
Help

Voltage Source AC

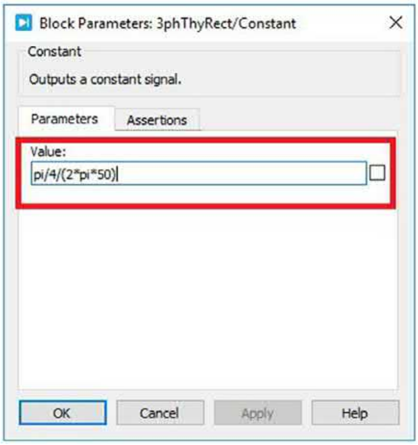
Generate a sinusoidal voltage.

Amplitude:
311 ☑

Frequency (rad/sec):
2*pi*50 ☑

Phase (rad):
pi/4 ☐

FIG. 3.134 Phase (rad) assumed to be $\frac{\pi}{4}$ in this example.


Block Parameters: 3phThyRect/Constant
Parameters
Assertions
OK
Cancel
Apply
Help

Constant

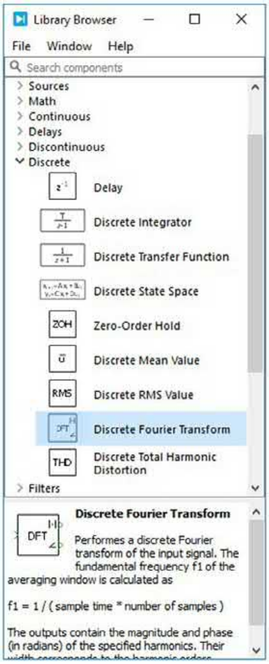
Outputs a constant signal.

Value: pi/4/(2*pi*50)

FIG. 3.135 “Constant” block settings (see Fig. 3.133).

3.4.2 Calculating the average output voltage using the “Discrete Fourier transform” block

We studied different methods of calculating the mean value of a waveform before. Yet there is another method: using the “Discrete Fourier Transform” block. “Discrete Fourier Transform” block measures the magnitudes/phase of harmonics specified by the user. If you specify that you want the magnitude of 0th harmonic, it calculates the mean value for you (Fig. 3.136).


Library Browser
File
Window
Help
Search components
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Delay
Discrete Integrator
Discrete Transfer Function
Discrete State Space
Zero-Order Hold
Discrete Mean Value
Discrete RMS Value
Discrete Fourier Transform
Discrete Total Harmonic Distortion
Filters
DFT

Discrete Fourier Transform

Performes a discrete Fourier transform of the input signal. The fundamental frequency f1 of the averaging window is calculated as

f1 = 1 / (sample time * number of samples)

The outputs contain the magnitude and phase (in radians) of the specified harmonics. Their

FIG. 3.136 “Discrete Fourier Transform” block.

For example, assume that we want to calculate the average value (mean value) of output voltage of full-wave single-phase controlled rectifier studied before. We change the schematic to that shown in Fig. 3.137.

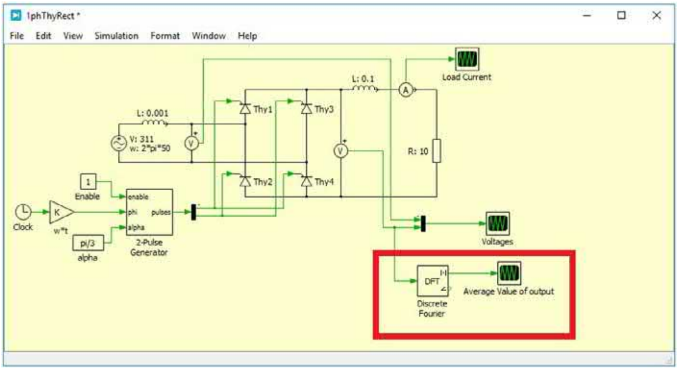
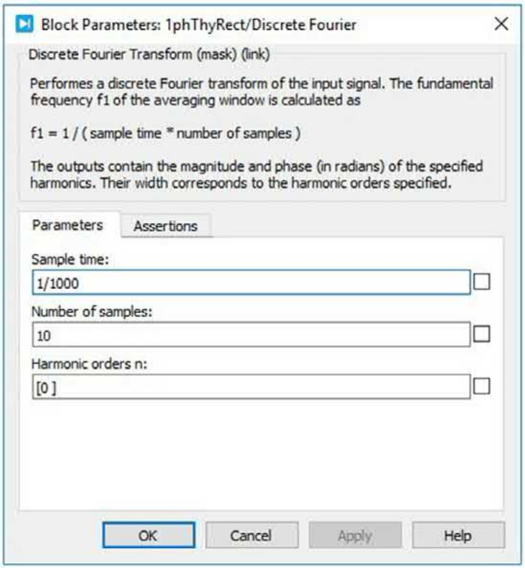

1phThyRect *
File Edit View Simulation Format Window Help
L: 0.1
Load Current
Thy1
Thy3
L: 0.001
V: 311
w: 2*pi*50
R: 10
Thy2
Thy4
1
Enable
enable
phi
pulses
alpha
2-Pulse Generator
Clock
K
w*t
pi/3
alpha
Voltages
DFT
Discrete Fourier
Average Value of output

FIG. 3.137 Adding the "Discrete Fourier Transform" block to the schematic.

Double click the "Discrete Fourier" block to open it and do the settings as shown in Fig. 3.138.


Block Parameters: 1phThyRect/Discrete Fourier

Discrete Fourier Transform (mask) (link)

Performes a discrete Fourier transform of the input signal. The fundamental frequency f1 of the averaging window is calculated as

f1 = 1 / (sample time * number of samples)

The outputs contain the magnitude and phase (in radians) of the specified harmonics. Their width corresponds to the harmonic orders specified.

Sample time: 1/1000

Number of samples: 10

Harmonic orders n: [0]

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.138 "Discrete Fourier Transform" block settings.

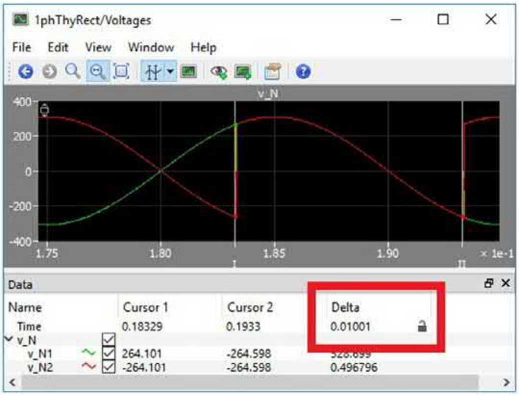
As shown in Fig. 3.138, the fundamental frequency of "Discrete Fourier Transform" block is calculated with the aid of:

$$f = \frac{1}{\text{Sample time} \times \text{Number of samples}}$$

Since we enter the “Sample time:” box as $\frac{1}{1000}$ and “Number of samples:” as 10, the fundamental frequency is calculated as:

$$f = \frac{1}{\frac{1}{1000} \times 10} = 100\,\mathrm{Hz}$$

“Number of samples” can be selected as 10 for most of simulations, but you can increase it if you need more accuracy. For example if “Number of samples” is increased to 20, “Sample time:” must be set to 0.0005 to keep the fundamental frequency constant at 100 Hz. Remember that full-wave rectifier output voltage has frequency two times greater than input source frequency. You can verify this issue by capturing a period of output voltage between two cursors and measure the time difference (Delta) between the cursors. As shown in Fig. 3.139, the period length is about .01.So, frequency is $\frac{1}{0.01} = 100\,\mathrm{Hz}$.


1phThyRect/Voltages
File
Edit
View
Window
Help
v_N
400
200
0
-200
-400
1.75
1.80
1.85
1.90
× 1e-1
I
II
Data

Name		Cursor 1	Cursor 2	Delta
Time		0.18329	0.1933	0.01001
v_N	☑			
v_N1	☑	264.101	-264.598	528.699
v_N2	☑	-264.101	-264.598	0.496796

FIG. 3.139 One period of load voltage takes about 0.01 s. So, its frequency is about 100 Hz.

If we run the simulation, we obtain the waveform shown in Fig. 3.140.

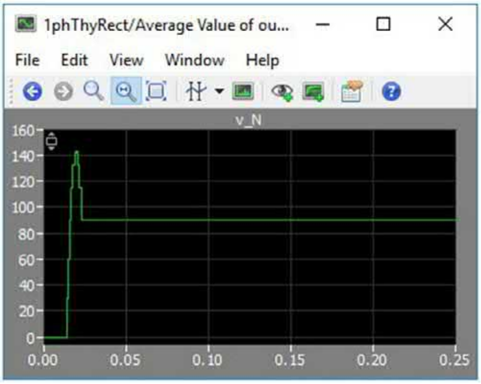

1phThyRect/Average Value of ou...
File
Edit
View
Window
Help
v_N
160
140
120
100
80
60
40
20
0
0.00
0.05
0.10
0.15
0.20
0.25

FIG. 3.140 Average value of load voltage.

Zoom in the waveform to obtain a better view (Fig. 3.141).

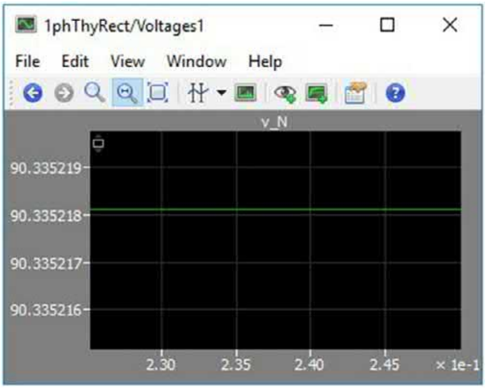
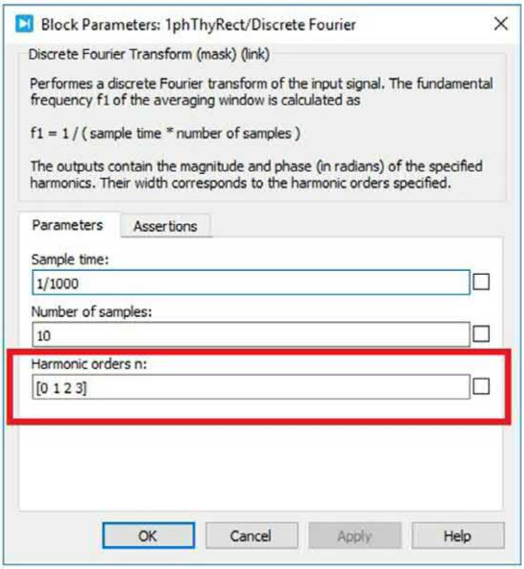


FIG. 3.141 Average value of load voltage is about 90.3 V.

If you want to measure other harmonics in the load voltage, double click on the "Discrete Fourier" block and enter the wanted harmonic into the "Harmonic orders n:" box. For example if we need harmonics up to 3rd one, we fill the "Harmonic orders n:" box as shown in Fig. 3.142. Simulation result is shown in Fig. 3.143.


Block Parameters: 1phThyRect/Discrete Fourier

Discrete Fourier Transform (mask) (link)

Performes a discrete Fourier transform of the input signal. The fundamental frequency f1 of the averaging window is calculated as

f1 = 1 / (sample time * number of samples)

The outputs contain the magnitude and phase (in radians) of the specified harmonics. Their width corresponds to the harmonic orders specified.

Sample time: 1/1000 ☐

Number of samples: 10 ☐

Harmonic orders n: [0 1 2 3] ☐

FIG. 3.142 Discrete Fourier block settings.

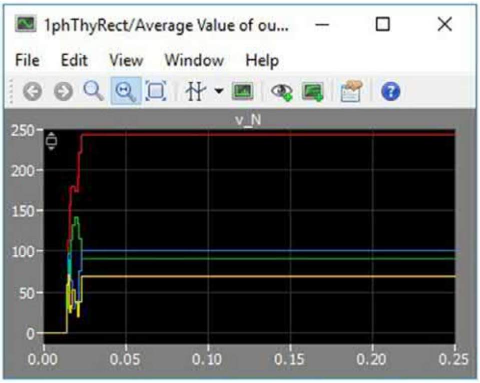


FIG. 3.143 PLECS shows the harmonics. DC component (green), first harmonic (red), second harmonic (blue), and third harmonic (yellow).

3.5 Example 3.5: 3 Phase full-wave controlled rectifier

In this example, we will simulate a 3-phase full-wave rectifier. The output voltages magnitude can be controlled using the thyristor firing angle.

3.5.1 Preparing the simulation

Fig. 3.144 shows a 3-phase controlled rectifier. The required gate pulses are provided by the "6-Pulse Generator" block (see Fig. 3.145). Some of the blocks settings used are shown in Figs. 3.146–3.148.

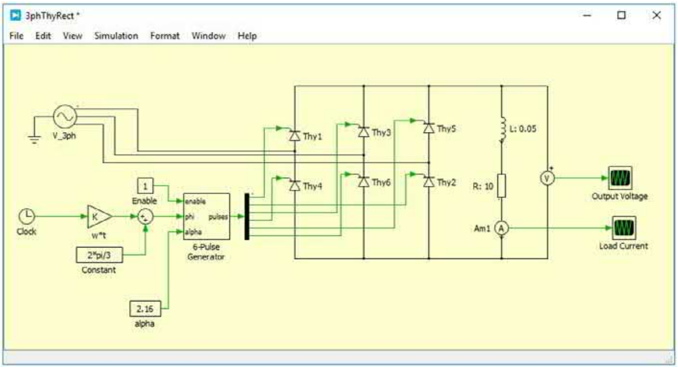
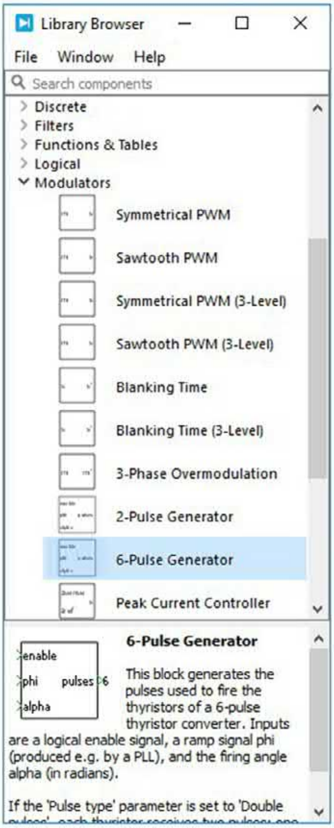

3phThyRect *
File
Edit
View
Simulation
Format
Window
Help
V_3ph
Thy1
Thy3
Thy5
Thy4
Thy6
Thy2
L: 0.05
R: 10
Am1
Output Voltage
Load Current
1
Enable
enable
phi
pulses
alpha
6-Pulse Generator
Clock
K
w*t
2*pi/3
Constant
2.16
alpha

FIG. 3.144 Controlled 3-phase rectifier.

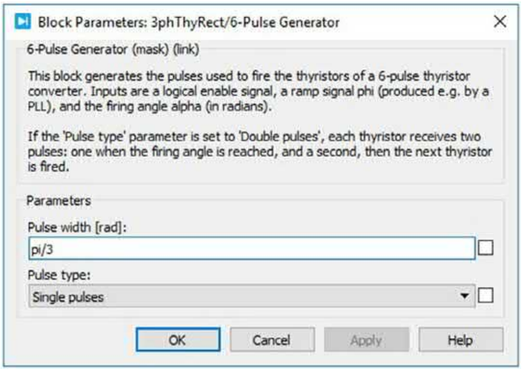

Library Browser
File Window Help
Search components
Discrete
Filters
Functions & Tables
Logical
Modulators
Symmetrical PWM
Sawtooth PWM
Symmetrical PWM (3-Level)
Sawtooth PWM (3-Level)
Blanking Time
Blanking Time (3-Level)
3-Phase Overmodulation
2-Pulse Generator
6-Pulse Generator
Peak Current Controller
enable
phi
alpha
pulses
6

6-Pulse Generator

This block generates the pulses used to fire the thyristors of a 6-pulse thyristor converter. Inputs are a logical enable signal, a ramp signal phi (produced e.g. by a PLL), and the firing angle alpha (in radians).

If the 'Pulse type' parameter is set to 'Double

FIG. 3.145 6-Pulse Generator block.


Block Parameters: 3phThyRect/6-Pulse Generator

6-Pulse Generator (mask) (link)

This block generates the pulses used to fire the thyristors of a 6-pulse thyristor converter. Inputs are a logical enable signal, a ramp signal phi (produced e.g. by a PLL), and the firing angle alpha (in radians).

If the 'Pulse type' parameter is set to 'Double pulses', each thyristor receives two pulses: one when the firing angle is reached, and a second, then the next thyristor is fired.

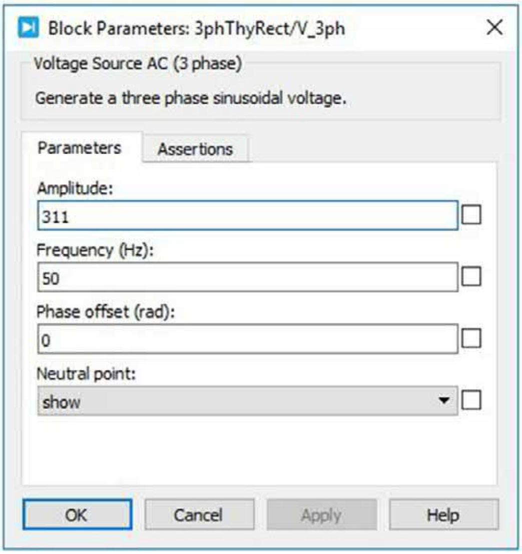
Parameters

Pulse width [rad]: pi/3 ☐

Pulse type: Single pulses ☐

OK
Cancel
Apply
Help

FIG. 3.146 6-Pulse generator block settings.


Block Parameters: 3phThyRect/V_3ph

Voltage Source AC (3 phase)

Generate a three phase sinusoidal voltage.

Amplitude: 311

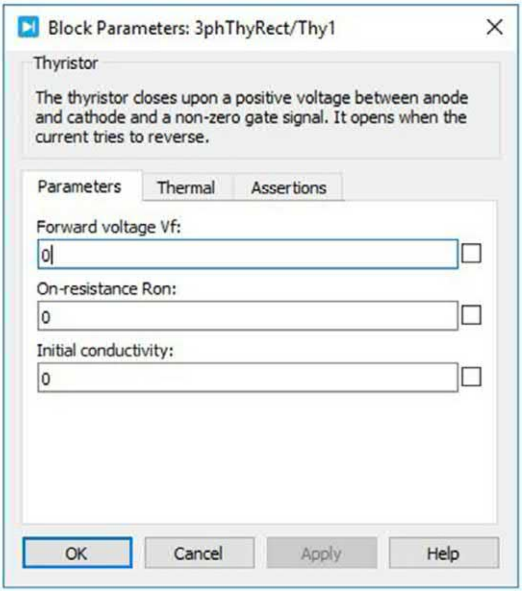
Frequency (Hz): 50

Phase offset (rad): 0

Neutral point: show

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.147 Input AC source block settings.


Block Parameters: 3phThyRect/Thy1

Thyristor

The thyristor closes upon a positive voltage between anode and cathode and a non-zero gate signal. It opens when the current tries to reverse.

Forward voltage Vf: 0

On-resistance Ron: 0

Initial conductivity: 0

Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

FIG. 3.148 All the six thyristors are set as shown in the figure.

If we run the simulation, we obtain the results shown in Figs. 3.149 and 3.150.

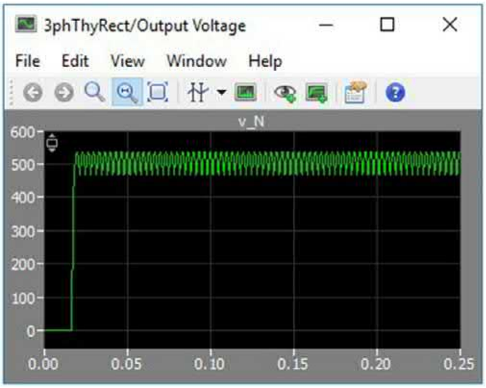

3phThyRect/Output Voltage
File
Edit
View
Window
Help
v_N
600
500
400
300
200
100
0
0.00
0.05
0.10
0.15
0.20
0.25

FIG. 3.149 Load voltage.

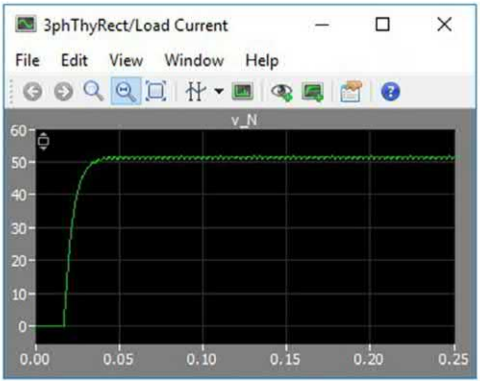
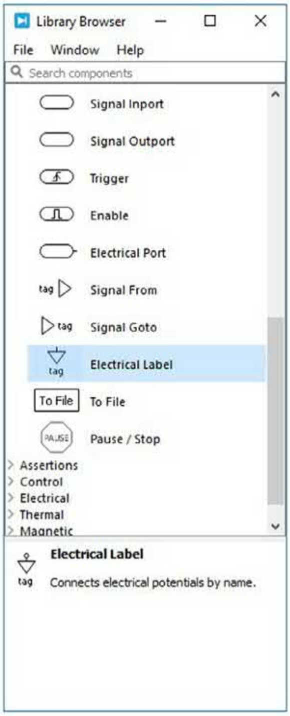

3phThyRect/Load Current
File
Edit
View
Window
Help
v_N
60
50
40
30
20
10
0
0.00
0.05
0.10
0.15
0.20
0.25

FIG. 3.150 Load current.

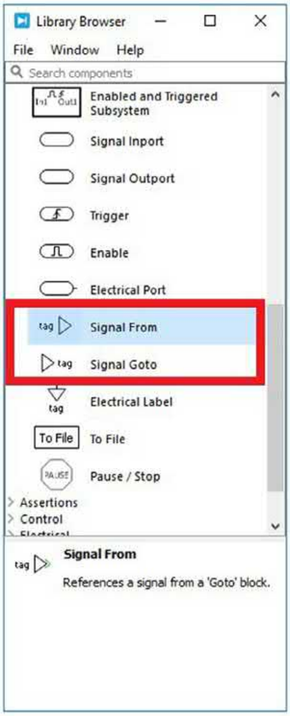
3.5.2 Drawing more understandable schematics using "Electrical label," "Signal from," and "Signal goto" blocks

Schematic shown in Fig. 3.144 seems crowded. We can draw it more understandable using the "Electrical Label," "Signal From," and "Signal Goto" blocks (Figs. 3.151 and 3.152).


Library Browser
File
Window
Help
Search components
Signal Inport
Signal Outport
Trigger
Enable
Electrical Port
tag
Signal From
tag
Signal Goto
tag
Electrical Label
To File
To File
PAUSE
Pause / Stop
Assertions
Control
Electrical
Thermal
Magnetic
tag
Electrical Label

Connects electrical potentials by name.

FIG. 3.151 Electrical Label block.


Library Browser
File
Window
Help
Search components
In1
Out1
Enabled and Triggered Subsystem
Signal Inport
Signal Outport
Trigger
Enable
Electrical Port
tag
Signal From
tag
Signal Goto
tag
Electrical Label
To File
To File
PAUSE
Pause / Stop
Assertions
Control
Electrical
tag
Signal From

References a signal from a 'Goto' block.

FIG. 3.152 "Signal From" and "Signal Goto" block.

Place 6 “Electrical Label” blocks in the schematic. Double click on them and change their “Tag name” to A, B, and C. So, you have two “Electrical Label” block with “Tag name” A, two “Electrical Label” block with “Tag name” B, and two “Electrical Label” block with “Tag name” C. Remove the wires between the source and thyristor legs and change the schematic as shown in Fig. 3.153. All the “Electrical Label” blocks with the same “Tag name”s are considered to be connected together. So, Fig. 3.153 is equal to Fig. 3.144 from PLECS viewpoint. But Fig. 3.153 is more understandable from user viewpoint.

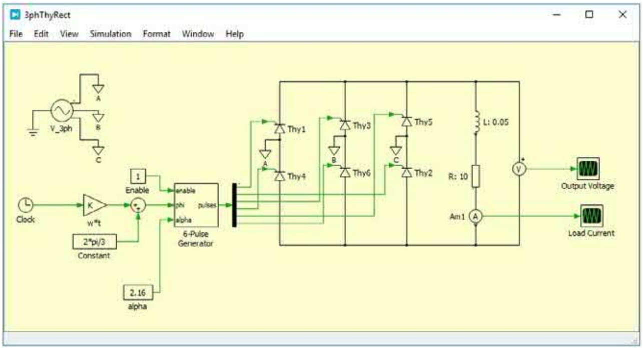

3phThyRect
File Edit View Simulation Format Window Help
V_3ph
A
B
C
Thy1
Thy3
Thy5
Thy4
Thy6
Thy2
L: 0.05
R: 10
Am1
Output Voltage
Load Current
1
Enable
enable
phi
pulses
alpha
6-Pulse Generator
Clock
K
w*t
2*pi/3
Constant
2.16
alpha

FIG. 3.153 Schematic is drawn using the “Electrical Label” blocks.

You can even simplify the schematic shown in Fig. 3.153 more as shown in Fig. 3.154.

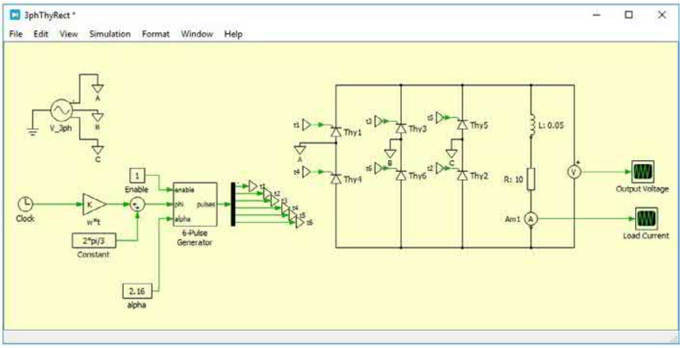

3phThyRect *
File Edit View Simulation Format Window Help
V_3ph
A
B
C
t1
t3
t5
t4
t6
t2
Thy1
Thy3
Thy5
Thy4
Thy6
Thy2
L: 0.05
R: 10
Am1
Output Voltage
Load Current
1
Enable
enable
phi
pulses
alpha
6-Pulse Generator
Clock
K
w*t
2*pi/3
Constant
2.16
alpha

FIG. 3.154 Schematic looks better when you use the label instead of connecting the parts directly together.

As shown in Fig. 3.153, the wire connecting the multiplexer block to the thyristor gates is green, so they are control signals and you cannot use “Electrical Label” since it is designed to work with power circuit components.

To connect the output of multiplexer to the thyristor gates, you must use "Signal From" and "Signal Goto" blocks.

Add a "Signal Goto" block to the schematic and connect it to first output of multiplexer block (Fig. 3.155).

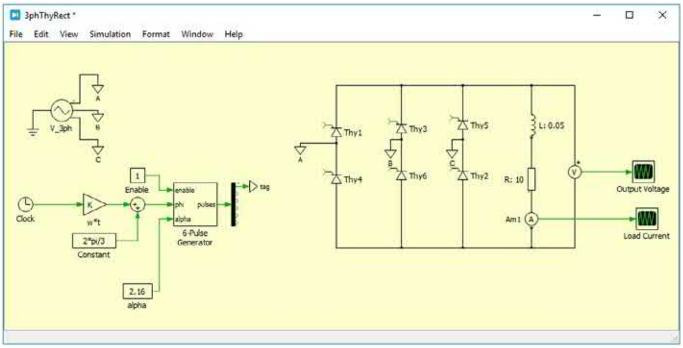
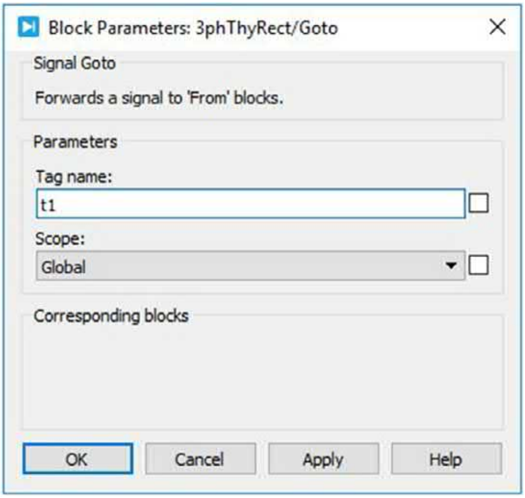

3phThyRect *
File Edit View Simulation Format Window Help
A
B
C
V_3ph
Clock
K
w*t
2*pi/3
Constant
1
Enable
enable
phi
pulses
alpha
6-Pulse Generator
2.16
alpha
tag
A
B
C
Thy1
Thy3
Thy5
Thy4
Thy6
Thy2
L: 0.05
R: 10
Am1
V
A
Output Voltage
Load Current

FIG. 3.155 Connect the "Signal Goto" to the first output of the multiplexer.

Double click on the "Signal Goto" block and set the "Tag name" to "t1" and click the OK button (Fig. 3.156).



Block Parameters: 3phThyRect/Goto

Signal Goto

Forwards a signal to 'From' blocks.

Parameters

Tag name: t1 ☐

Scope: Global ☐

Corresponding blocks

OK
Cancel
Apply
Help

FIG. 3.156 Renaming the placed "Signal Goto" block to "t1".

After setting the "Tag name" to "t1," the schematic looks like that shown in Fig. 3.157.

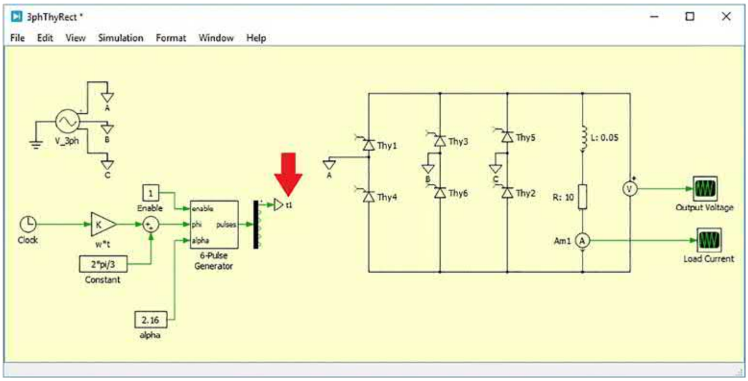

3phThyRect *
File Edit View Simulation Format Window Help
A
V_3ph
B
C
Clock
K
w*t
2*pi/3
Constant
1
Enable
enable
phi
alpha
pulses
6-Pulse Generator
2.16
alpha
t1
A
Thy1
Thy4
Thy3
Thy6
B
Thy5
Thy2
C
L: 0.05
R: 10
V
Am1
A
Output Voltage
Load Current

FIG. 3.157 After renaming the block, new name (t1) will appear on the schematic.

Place a “Signal From” block on the schematic and connect it to the gate of the thyristor named “Thy1” (Fig. 3.158).

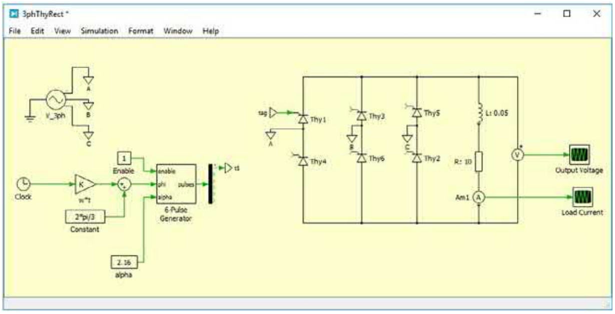
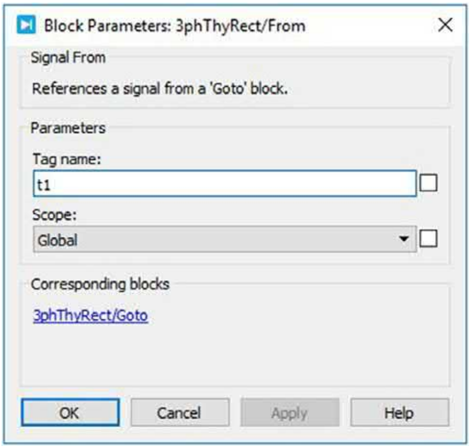

3phThyRect *
File Edit View Simulation Format Window Help
A
V_3ph
B
C
Clock
K
w*t
2*pi/3
Constant
1
Enable
enable
phi
alpha
pulses
6-Pulse Generator
2.16
alpha
t1
tag
A
Thy1
Thy4
Thy3
Thy6
B
Thy5
Thy2
C
L: 0.05
R: 10
V
Am1
A
Output Voltage
Load Current

FIG. 3.158 Connect the “Signal From” to the gate of thyristor “Thy1.”]

Double click the “Signal From” block and set the “Tag name:” to “t1” and click the OK button (Fig. 3.159).


Block Parameters: 3phThyRect/From

Signal From

References a signal from a 'Goto' block.

Parameters

Tag name: t1 ☐

Scope: Global ☐

Corresponding blocks

3phThyRect/Goto

OK
Cancel
Apply
Help

FIG. 3.159 Renaming the placed “Signal From” block to “t1”.

After setting the “Tag name” to “t1,” the schematic looks like that shown in Fig. 3.160.

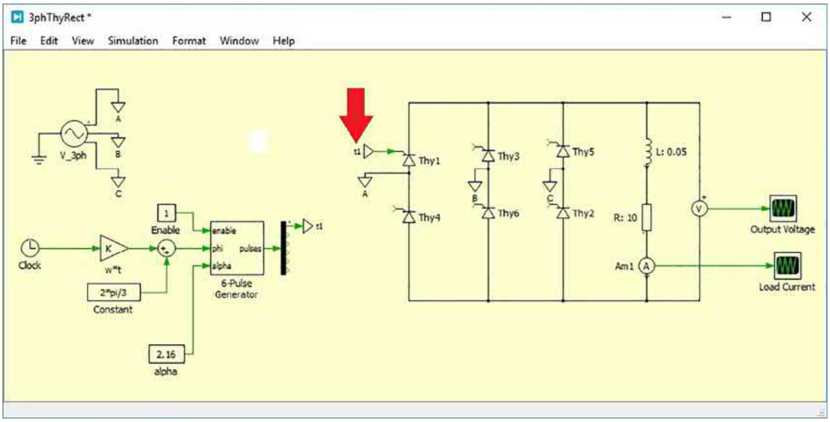

3phThyRect *
File Edit View Simulation Format Window Help
A
B
C
V_3ph
Clock
K
w*t
2*pi/3
Constant
1
Enable
enable
phi
pulses
alpha
6-Pulse Generator
2.16
alpha
t1
t1
A
B
C
Thy1
Thy4
Thy3
Thy6
Thy5
Thy2
L: 0.05
R: 10
Am1
A
V
Output Voltage
Load Current

FIG. 3.160 After renaming the block, new name (t1) will appear on the schematic.

Do the same procedure to place 5 more “Signal Goto” and “Signal From” blocks on the schematic. Name them t2, t3, t4, t5, and t6. Do the required connection in order to change your schematic into the one shown in Fig. 3.154. All the “Signal Goto” and “Signal From” blocks with the same “Tag name” are considered to be connected.

3.5.3 Delay block

Assume a rectifier shown in Fig. 3.161.

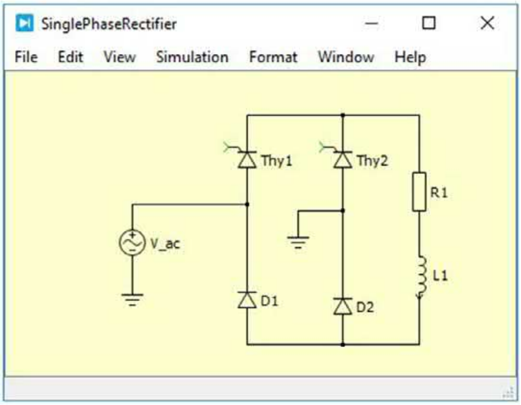
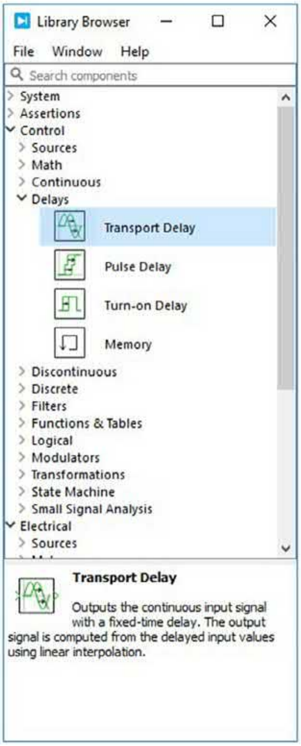

SinglePhaseRectifier
File Edit View Simulation Format Window Help
Thy1
Thy2
R1
V_ac
L1
D1
D2

FIG. 3.161 Another topology of controlled single-phase rectifiers.

In this type of rectifier, control signal for "Thy2" is a delayed version of gate pulse applied to gate of "Thy1." So, there is no need to generate a separate gate signal for "Thy2." You can use the delayed version of the pulse send to the gate of "Thy1." PLECS has a block named "Transport Delay," which can be used for such situations (Fig. 3.162).

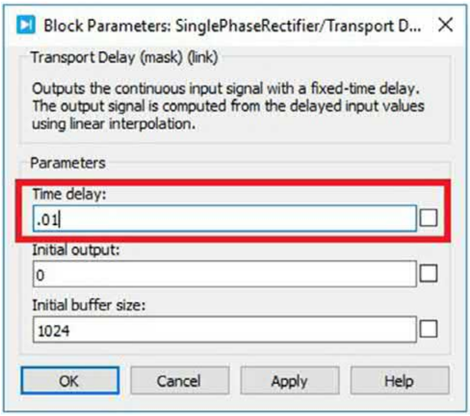

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Transport Delay
Pulse Delay
Turn-on Delay
Memory
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Transformations
State Machine
Small Signal Analysis
Electrical
Sources

Transport Delay

Outputs the continuous input signal with a fixed-time delay. The output signal is computed from the delayed input values using linear interpolation.

FIG. 3.162 Transport Delay block.

You can set the amount of required delay by the "Time delay:" box (Fig. 3.163).


Block Parameters: SinglePhaseRectifier/Transport D...
OK
Cancel
Apply
Help

Transport Delay (mask) (link)

Outputs the continuous input signal with a fixed-time delay. The output signal is computed from the delayed input values using linear interpolation.

Parameters

Time delay: .01 ☐

Initial output: 0 ☐

Initial buffer size: 1024 ☐

FIG. 3.163 Setting the required time delay.

3.6 Example 3.6: Boost converter

In this example, we will simulate a PWM step up converter.

3.6.1 Preparing the simulation

Fig. 3.164 shows the schematic of a Boost converter.

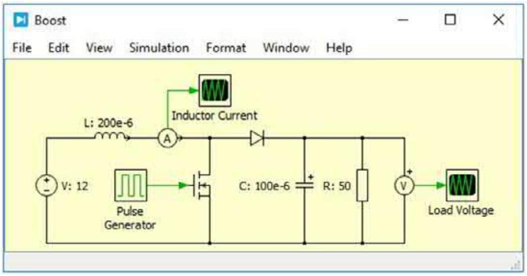
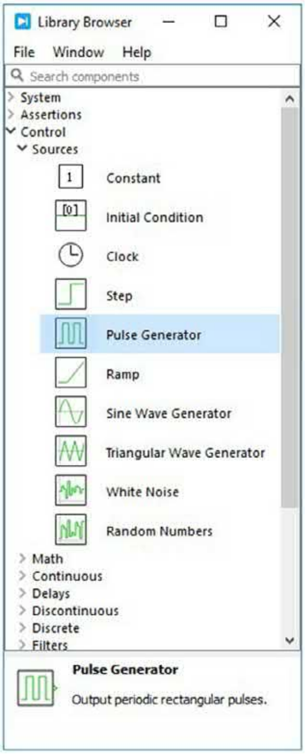

Boost
File
Edit
View
Simulation
Format
Window
Help
Inductor Current
L: 200e-6
A
V: 12
Pulse Generator
C: 100e-6
R: 50
V
Load Voltage

FIG. 3.164 Boost converter.

As shown in Fig. 3.164, the gate pulse is provided by a "Pulse Generator" block. "Pulse Generator" block provides a pulse with duty ratio of 0.6 and frequency of 25 KHz (Fig. 3.165).

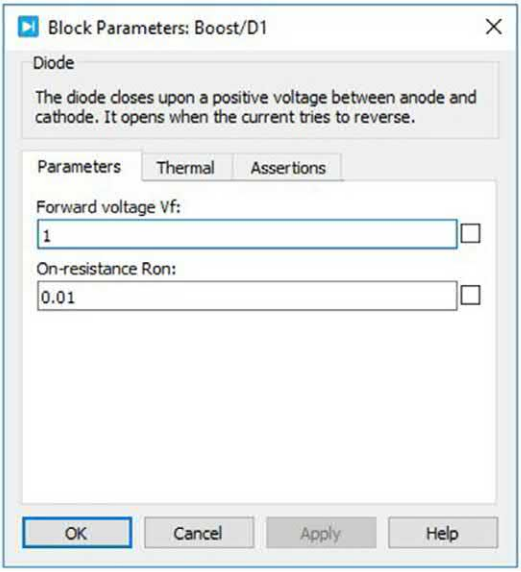

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
1
Constant
[0]
Initial Condition
Clock
Step
Pulse Generator
Ramp
Sine Wave Generator
Triangular Wave Generator
White Noise
Random Numbers
Math
Continuous
Delays
Discontinuous
Discrete
Filters

Pulse Generator

Output periodic rectangular pulses.

FIG. 3.165 Pulse Generator block.

Figs. 3.166–3.168 shows the blocks settings used. The MOSFET on resistance is assumed to be 40 mΩ. Diode voltage drop and series resistance are assumed to be 1 V and 0.01 Ω, respectively.


Block Parameters: Boost/D1

Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

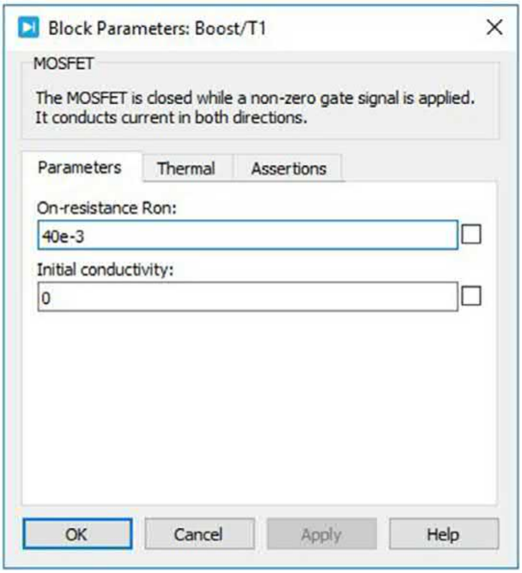
Parameters
Thermal
Assertions

Forward voltage Vf: 1 ☐

On-resistance Ron: 0.01 ☐

OK
Cancel
Apply
Help

FIG. 3.166 Diode settings.


Block Parameters: Boost/T1

MOSFET

The MOSFET is closed while a non-zero gate signal is applied. It conducts current in both directions.

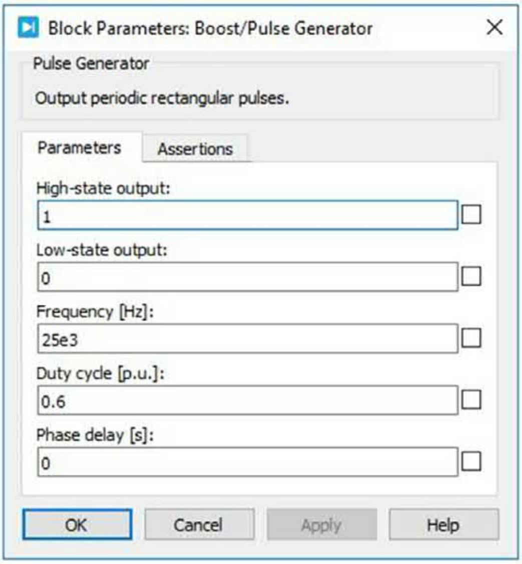
Parameters
Thermal
Assertions

On-resistance Ron: 40e-3 ☐

Initial conductivity: 0 ☐

OK
Cancel
Apply
Help

FIG. 3.167 MOSFET settings.


Block Parameters: Boost/Pulse Generator

Pulse Generator

Output periodic rectangular pulses.

Parameters
Assertions

High-state output: 1 ☐

Low-state output: 0 ☐

Frequency [Hz]: 25e3 ☐

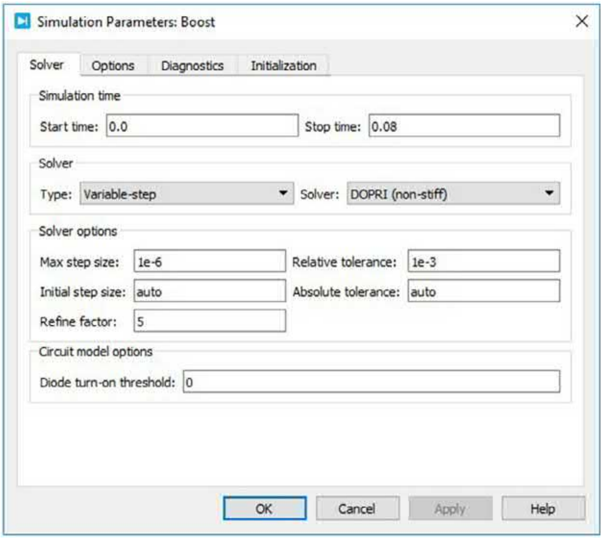
Duty cycle [p.u.]: 0.6 ☐

Phase delay [s]: 0 ☐

OK
Cancel
Apply
Help

FIG. 3.168 Pulse Generator settings.

If we run the simulation with the settings shown in Fig. 3.169, we obtain the results shown in Figs. 3.170–3.173.


Simulation Parameters: Boost

Solver
Options
Diagnostics
Initialization

Simulation time

Start time: 0.0

Stop time: 0.08

Solver

Type: Variable-step

Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-6

Relative tolerance: 1e-3

Initial step size: auto

Absolute tolerance: auto

Refine factor: 5

Circuit model options

Diode turn-on threshold: 0

OK
Cancel
Apply
Help

FIG. 3.169 Simulation parameter.

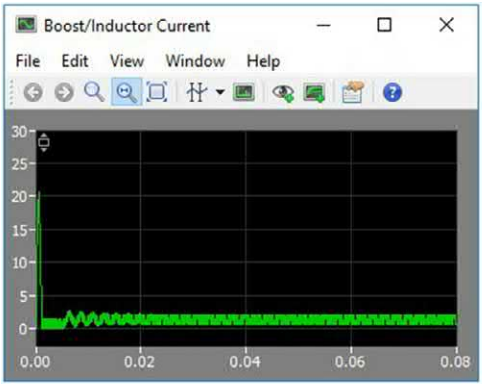

Boost/Inductor Current
File Edit View Window Help
30
25
20
15
10
5
0
0.00
0.02
0.04
0.06
0.08

FIG. 3.170 Inductor current waveform.

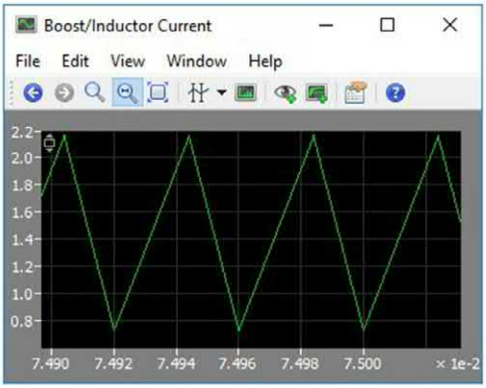

Boost/Inductor Current
File Edit View Window Help
2.2
2.0
1.8
1.6
1.4
1.2
1.0
0.8
7.490
7.492
7.494
7.496
7.498
7.500
× 1e-2

FIG. 3.171 Zooming in the inductor current waveform.

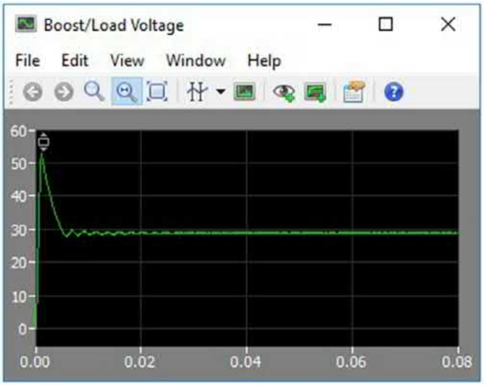

Boost/Load Voltage
File Edit View Window Help
60
50
40
30
20
10
0
0.00
0.02
0.04
0.06
0.08

FIG. 3.172 Load voltage waveform.

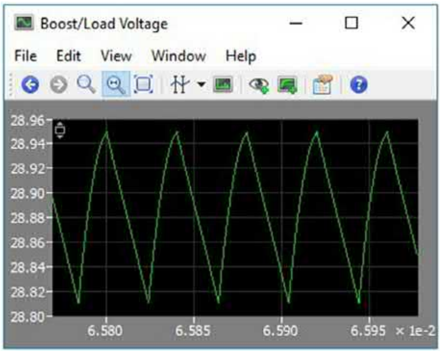
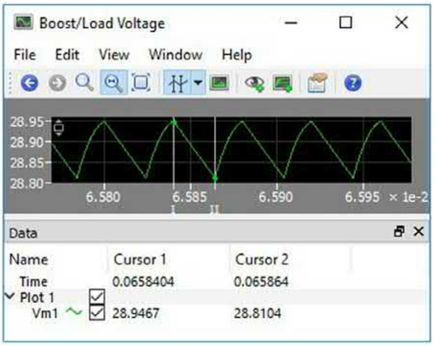

Boost/Load Voltage
File
Edit
View
Window
Help
28.96
28.94
28.92
28.90
28.88
28.86
28.84
28.82
28.80
6.580
6.585
6.590
6.595
× 1e-2

FIG. 3.173 Zooming in the load voltage.

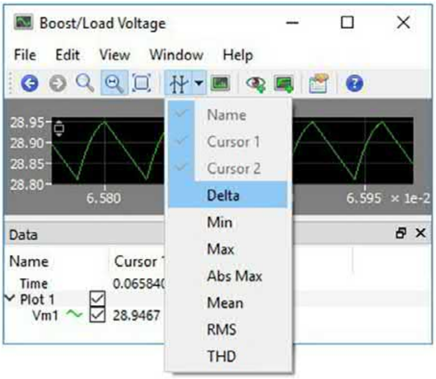
You can measure the output voltage ripple by placing a cursor on top of the curve and the other one on the bottom of the curve (Fig. 3.174).


Boost/Load Voltage
File
Edit
View
Window
Help
28.95
28.90
28.85
28.80
6.580
6.585
6.590
6.595
× 1e-2
I
II
Data

Name		Cursor 1	Cursor 2
Time		0.0658404	0.065864
Plot 1	☑		
Vm1	☑	28.9467	28.8104

FIG. 3.174 Putting the cursors on the maximum and minimum points to obtain the output voltage ripple.

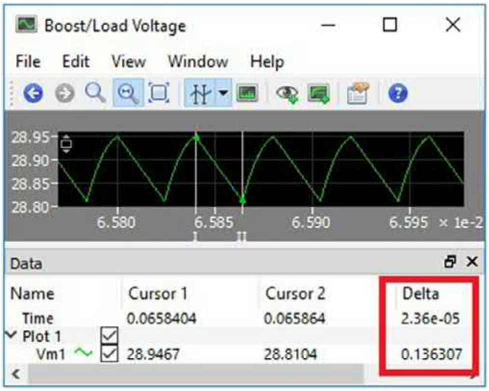
Click the small triangle behind the cursor icon and select "Delta" (Fig. 3.175)


Boost/Load Voltage
File
Edit
View
Window
Help
28.95
28.90
28.85
28.80
6.580
6.595
× 1e-2
Name
Cursor 1
Cursor 2
Delta
Min
Max
Abs Max
Mean
RMS
THD
Data

Name		Cursor
Time		0.06584
Plot 1	☑	
Vm1	☑	28.9467

FIG. 3.175 Measuring the difference between the cursors.

PLECS calculates the time difference and amplitude difference between the two cursors and shows it. So, the output voltage ripple is about 0.136 V = 136 mV (Fig. 3.176).


Boost/Load Voltage
File
Edit
View
Window
Help
28.95
28.90
28.85
28.80
6.580
6.585
6.590
6.595
× 1e-2
I
II
Data

Name		Cursor 1	Cursor 2	Delta
Time		0.0658404	0.065864	2.36e-05
Plot 1	☑			
Vm1	☑	28.9467	28.8104	0.136307

FIG. 3.176 The time and voltage difference between the two cursors is 2.36×10^{-5} s and 0.136 V, respectively.

3.6.2 Simulating the circuit using the ready-to-use modulator

Schematic shown in Fig. 3.164 uses the "Pulse Generator" block to turn the MOSFET on and off. Schematic shown in Fig. 3.177 uses the "Sawtooth PWM" block (see Fig. 3.178) to control the MOSFET. "Sawtooth PWM" block settings are shown in Fig. 3.179.

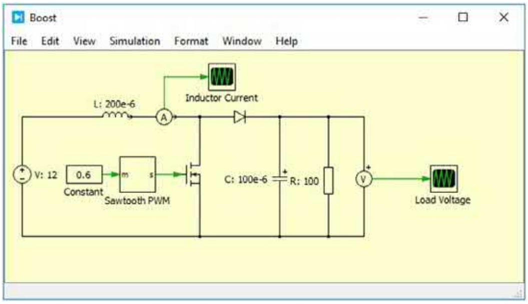
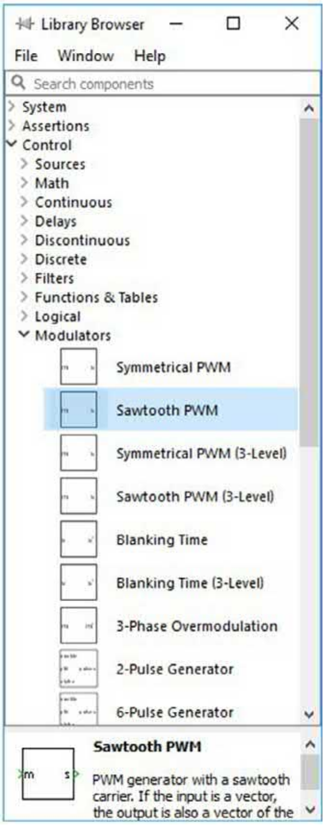

Boost
File
Edit
View
Simulation
Format
Window
Help
Inductor Current
L: 200e-6
A
V: 12
0.6
Constant
m
s
Sawtooth PWM
C: 100e-6
R: 100
V
Load Voltage

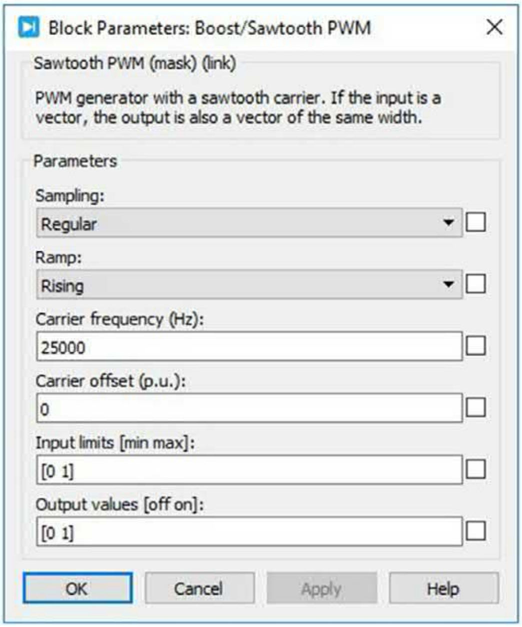
FIG. 3.177 Using the Sawtooth PWM block to turn on and off the MOSFET.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Symmetrical PWM
Sawtooth PWM
Symmetrical PWM (3-Level)
Sawtooth PWM (3-Level)
Blanking Time
Blanking Time (3-Level)
3-Phase Overmodulation
2-Pulse Generator
6-Pulse Generator
m
s

Sawtooth PWM

PWM generator with a sawtooth carrier. If the input is a vector, the output is also a vector of the

FIG. 3.178 Sawtooth PWM block.


Block Parameters: Boost/Sawtooth PWM
OK
Cancel
Apply
Help

Sawtooth PWM (mask) (link)

PWM generator with a sawtooth carrier. If the input is a vector, the output is also a vector of the same width.

Parameters

Sampling: Regular ☐

Ramp: Rising ☐

Carrier frequency (Hz): 25000 ☐

Carrier offset (p.u.): 0 ☐

Input limits [min max]: [0 1] ☐

Output values [off on]: [0 1] ☐

FIG. 3.179 Sawtooth PWM block settings.

3.6.3 Efficiency measurement

You can use the schematic shown in Fig. 3.180 to calculate the converter efficiency. This schematic uses a 10-Ω load. The schematic uses saturation, divide/product, and moving average blocks. These block places are shown in Figs. 3.181–3.183. Some of the blocks settings used are shown in Figs. 3.184–3.186. Here we compute the input and output power and divide them to obtain the efficiency.

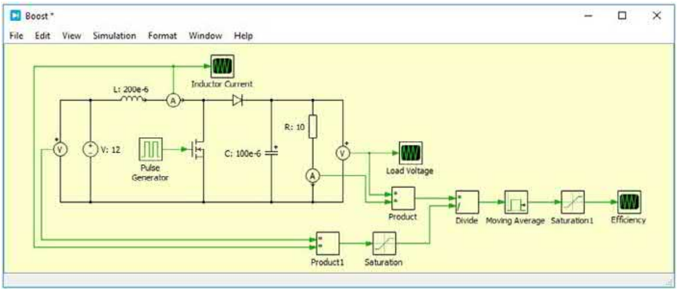
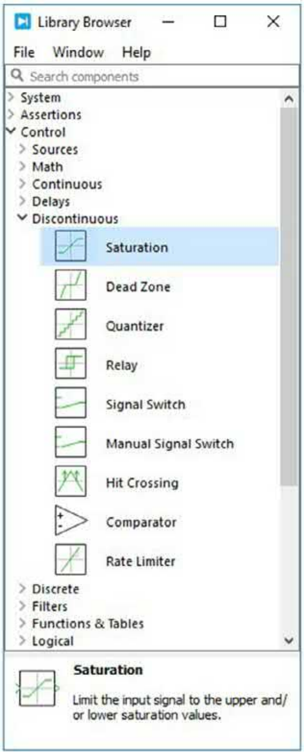

Boost *
File Edit View Simulation Format Window Help
Inductor Current
L: 200e-6
A
R: 10
V: 12
V
Pulse Generator
C: 100e-6
V
A
Load Voltage
Product
Divide
Moving Average
Saturation1
Efficiency
Product1
Saturation

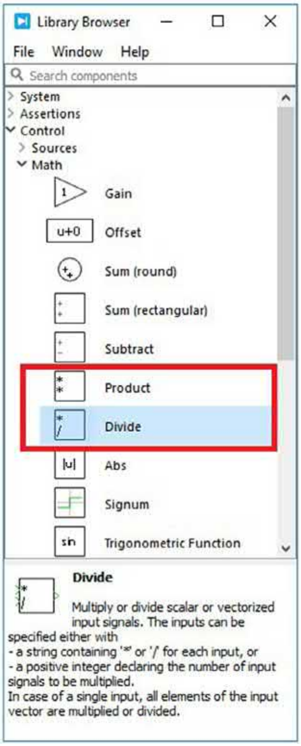
FIG. 3.180 Measuring the efficiency.


Library Browser
File Window Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Saturation
Dead Zone
Quantizer
Relay
Signal Switch
Manual Signal Switch
Hit Crossing
Comparator
Rate Limiter
Discrete
Filters
Functions & Tables
Logical

Saturation

Limit the input signal to the upper and/or lower saturation values.

FIG. 3.181 Saturation block.

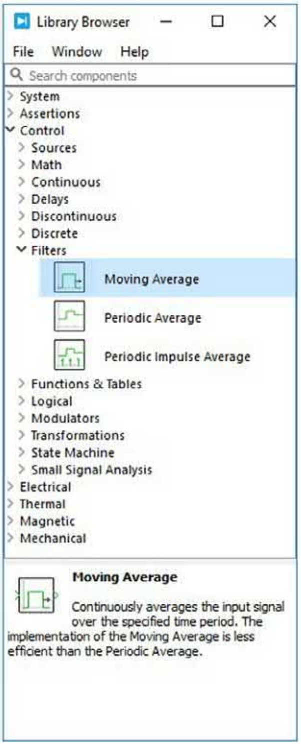

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Gain
Offset
Sum (round)
Sum (rectangular)
Subtract
Product
Divide
Abs
Signum
Trigonometric Function

Divide

Multiply or divide scalar or vectorized input signals. The inputs can be specified either with
- a string containing '*' or '/' for each input, or
- a positive integer declaring the number of input signals to be multiplied.

In case of a single input, all elements of the input vector are multiplied or divided.

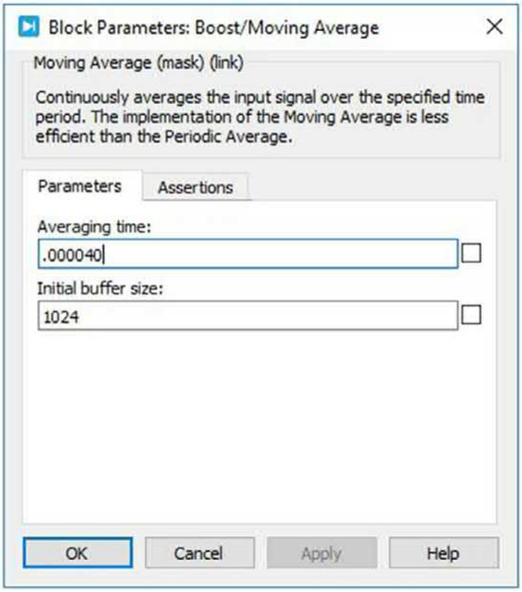
FIG. 3.182 "Product" and "Divide" blocks.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Moving Average
Periodic Average
Periodic Impulse Average
Functions & Tables
Logical
Modulators
Transformations
State Machine
Small Signal Analysis
Electrical
Thermal
Magnetic
Mechanical

Moving Average

Continuously averages the input signal over the specified time period. The implementation of the Moving Average is less efficient than the Periodic Average.

FIG. 3.183 "Moving Average" block.


Block Parameters: Boost/Moving Average

Moving Average (mask) (link)

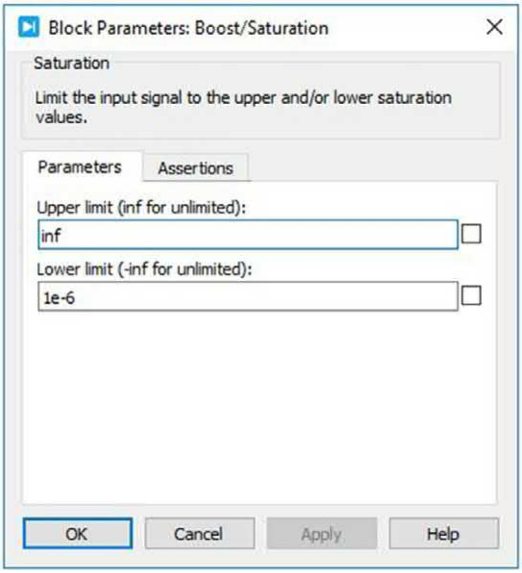
Continuously averages the input signal over the specified time period. The implementation of the Moving Average is less efficient than the Periodic Average.

Averaging time: .000040 ☐

Initial buffer size: 1024 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.184 "Moving Average" block settings (see Fig. 3.180). "Averaging time" must be set equal to the block input waveform period. Since the input has frequency of 25 KHz, "Averaging time" set to 40 μs.


Block Parameters: Boost/Saturation

Saturation

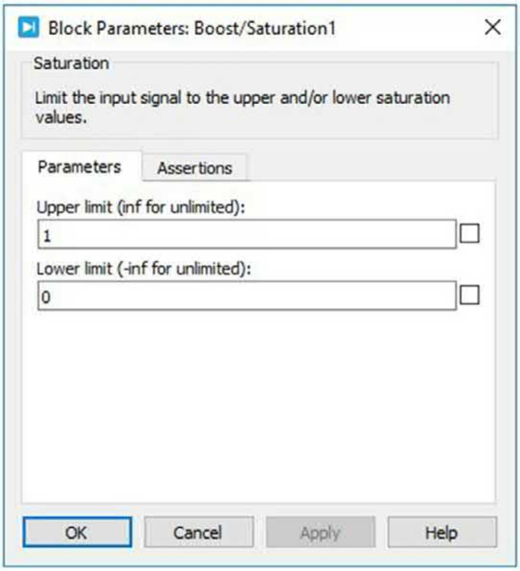
Limit the input signal to the upper and/or lower saturation values.

Upper limit (inf for unlimited): inf ☐

Lower limit (-inf for unlimited): 1e-6 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.185 "Saturation" block settings (see Fig. 3.180).


Block Parameters: Boost/Saturation1

Saturation

Limit the input signal to the upper and/or lower saturation values.

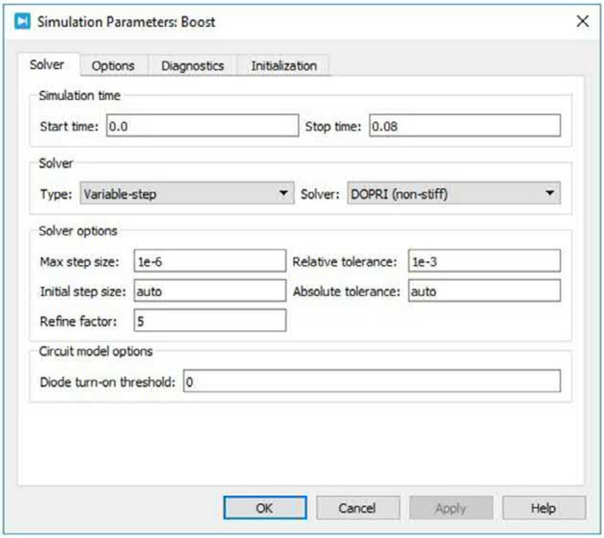
Upper limit (inf for unlimited): 1 ☐

Lower limit (-inf for unlimited): 0 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.186 "Saturation1" block settings (see Fig. 3.180).

If you run the simulation with settings shown in Fig. 3.187, you will obtain the results shown in Fig. 3.188.


Simulation Parameters: Boost

Simulation time

Start time: 0.0
Stop time: 0.08

Solver

Type: Variable-step
Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-6
Relative tolerance: 1e-3
Initial step size: auto
Absolute tolerance: auto
Refine factor: 5

Circuit model options

Diode turn-on threshold: 0

Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

FIG. 3.187 Simulation parameters.

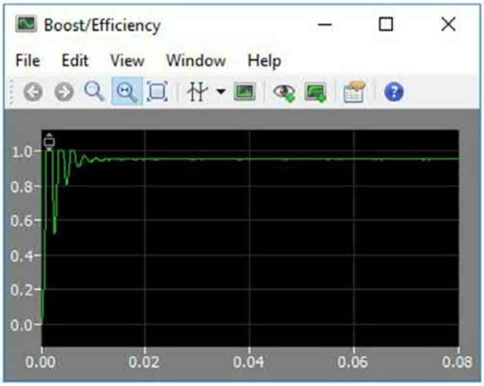

Boost/Efficiency
File Edit View Window Help
1.0
0.8
0.6
0.4
0.2
0.0
0.00
0.02
0.04
0.06
0.08

FIG. 3.188 Calculated efficiency.

You can zoom in to obtain a better view. As shown in Fig. 3.189, the efficiency is about 95%.

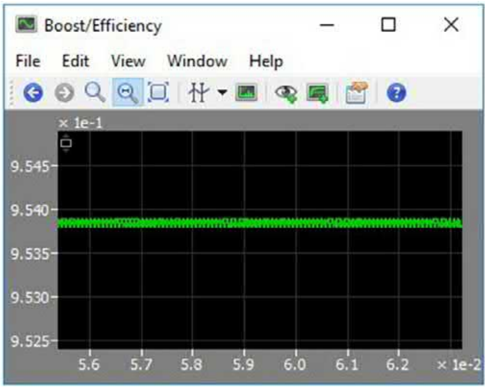

Boost/Efficiency
File Edit View Window Help
× 1e-1
9.545
9.540
9.535
9.530
9.525
5.6
5.7
5.8
5.9
6.0
6.1
6.2
× 1e-2

FIG. 3.189 Zooming the graph. When using the scope to monitor the efficiency, zoom into the steady state of graph. Transients part of graph is not important.

If you try to run the simulation without block named “Saturation”(see Fig. 3.190), you will face an error as shown in Fig. 3.191. The reason is: in $t=0$ the input and output power are zero, so PLECS faces $\frac{0}{0}$. In the schematic shown in Fig. 3.180, we set the lower bound for input power to be $1e-6=10^{-6}$ (see Fig. 3.185) instead of 0, so at $t=0$ PLECS faces $\frac{0}{10^{-6}}$, which causes no problem since it is equal to 0.

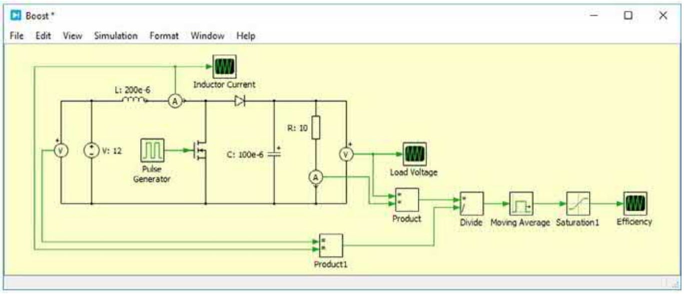
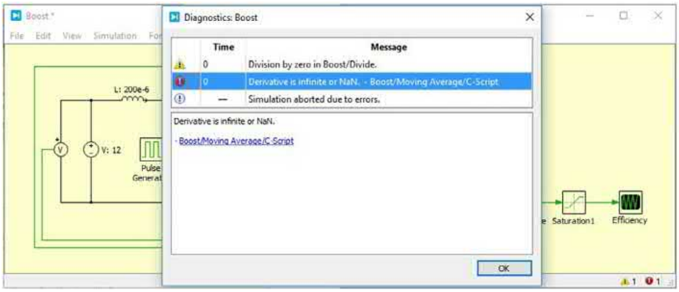

Boost *
File Edit View Simulation Format Window Help
L: 200e-6
Inductor Current
R: 10
V: 12
Pulse Generator
C: 100e-6
Load Voltage
Product
Divide
Moving Average
Saturation1
Efficiency
Product1

FIG. 3.190 Schematic diagram without “Saturation” block. Compare with Fig. 3.180.


Boost *
File Edit View Simulation Fo
L: 200e-6
V: 12
Pulse Generat
Saturation1
Efficiency
OK
1 1

Diagnostics: Boost

	Time	Message
⚠	0	Division by zero in Boost/Divide.
●	0	Derivative is infinite or NaN. - Boost/Moving Average/C-Script
ⓘ	—	Simulation aborted due to errors.

Derivative is infinite or NaN.

- Boost/Moving Average/C-Script

FIG. 3.191 Generated error message.

3.7 Example 3.7: Obtaining the small signal transfer functions for a buck converter

In this example, we will show how you can obtain the small signal transfer functions for a DC-DC step down converter. The obtained Bode plots are used to design the controller. See Chapter 7 of [1] to see an example.

3.7.1 Preparing the simulation

Assume a Buck converter schematic shown in Fig. 3.192. Some of the used blocks settings are shown in Figs. 3.193–3.195.

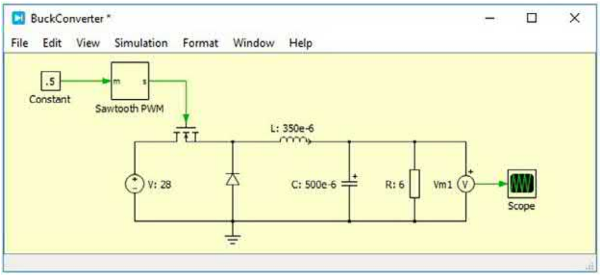
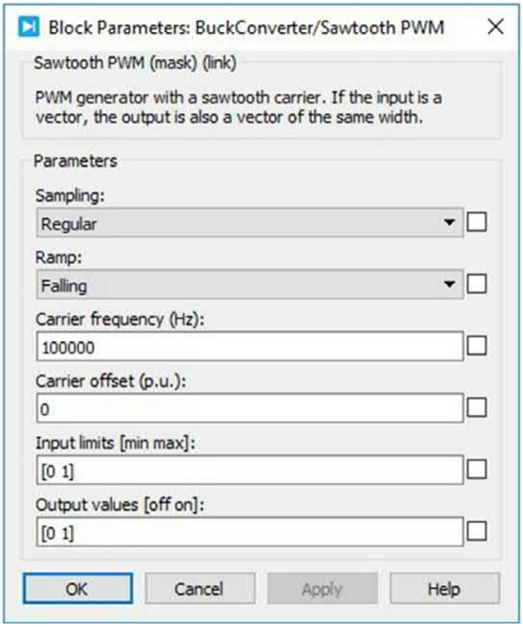

BuckConverter *
File Edit View Simulation Format Window Help
.5
Constant
m
s
Sawtooth PWM
L: 350e-6
V: 28
C: 500e-6
R: 6
Vm1
Scope

FIG. 3.192 Buck converter.



Block Parameters: BuckConverter/Sawtooth PWM

Sawtooth PWM (mask) (link)

PWM generator with a sawtooth carrier. If the input is a vector, the output is also a vector of the same width.

Parameters

Sampling: Regular ☐

Ramp: Falling ☐

Carrier frequency (Hz): 100000 ☐

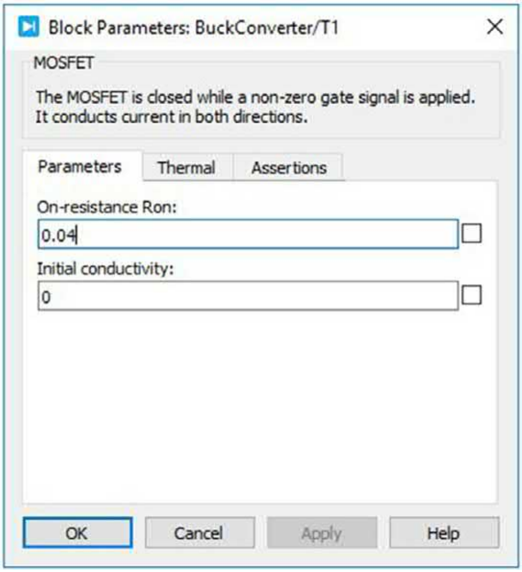
Carrier offset (p.u.): 0 ☐

Input limits [min max]: [0 1] ☐

Output values [off on]: [0 1] ☐

OK
Cancel
Apply
Help

FIG. 3.193 Sawtooth PWM block settings.



Block Parameters: BuckConverter/T1

MOSFET

The MOSFET is closed while a non-zero gate signal is applied. It conducts current in both directions.

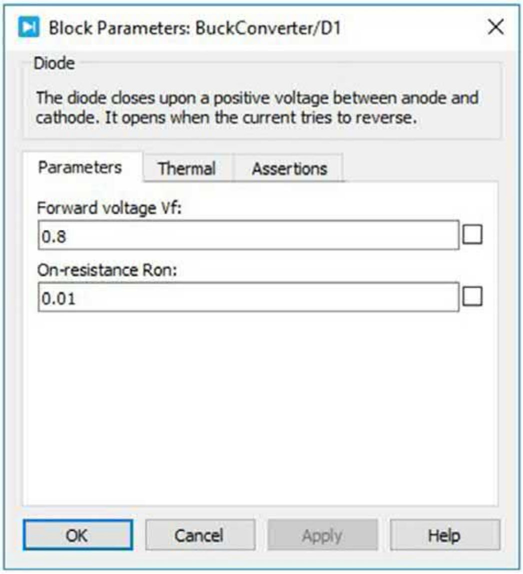
Parameters
Thermal
Assertions

On-resistance Ron: 0.04 ☐

Initial conductivity: 0 ☐

OK
Cancel
Apply
Help

FIG. 3.194 MOSFET settings.


Block Parameters: BuckConverter/D1

Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

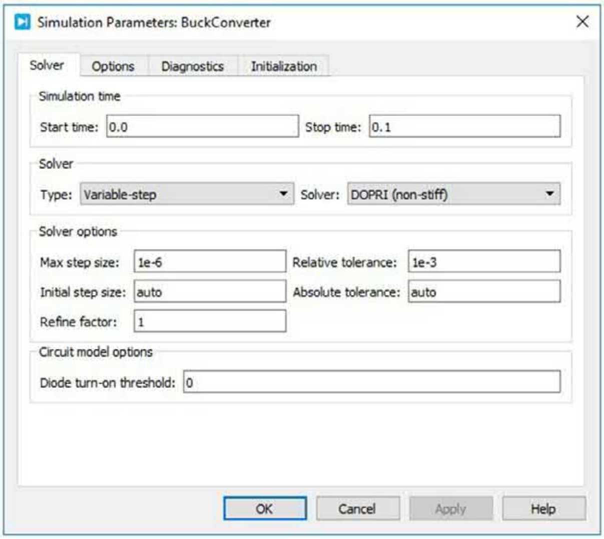
Forward voltage Vf: 0.8

On-resistance Ron: 0.01

Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

FIG. 3.195 Diode settings.

If we run the simulation with settings shown in Fig. 3.196, we obtain the waveform shown in Fig. 3.197. Output voltage is about 14 V.


Simulation Parameters: BuckConverter

Simulation time

Start time: 0.0

Stop time: 0.1

Solver

Type: Variable-step

Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-6

Relative tolerance: 1e-3

Initial step size: auto

Absolute tolerance: auto

Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

FIG. 3.196 Simulation Parameters.

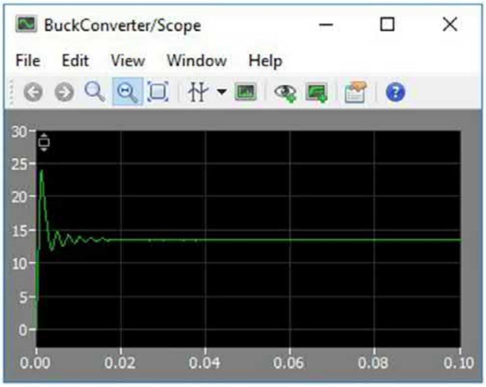

BuckConverter/Scope
File Edit View Window Help
30
25
20
15
10
5
0
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 3.197 Output voltage waveform.

Assume that we want to obtain the small signal transfer function from control input (duty ratio) to output voltage. To do this, we change the schematic as shown in Fig. 3.198. This schematic uses "Perturbation" and "Response" blocks. These blocks can be found in the "Small Signal Analysis" section of Library Browser (see Fig. 3.199). "Perturbation" and "Response" blocks settings are shown in Figs. 3.200 and 3.201.

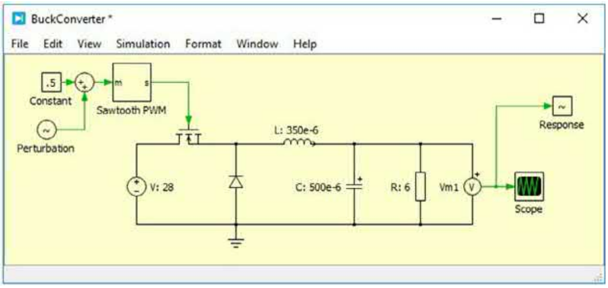
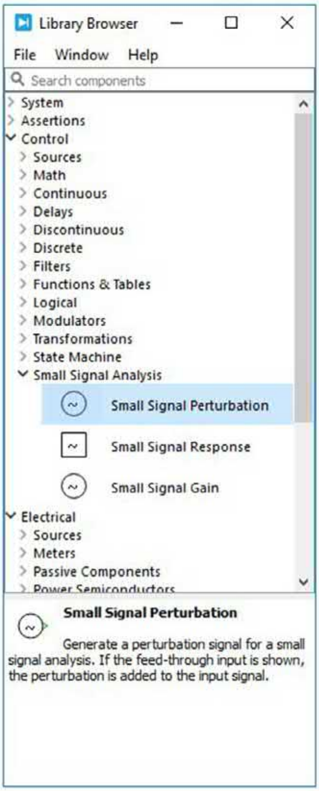

BuckConverter *
File Edit View Simulation Format Window Help
.5
Constant
Perturbation
m s
Sawtooth PWM
L: 350e-6
Response
V: 28
C: 500e-6
R: 6
Vm1
Scope

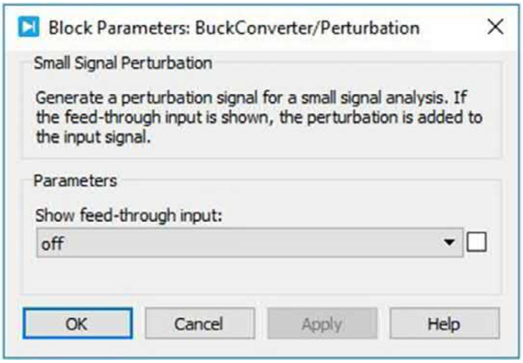
FIG. 3.198 Adding the "Perturbation" and "Response" blocks to the schematic.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Transformations
State Machine
Small Signal Analysis
Small Signal Perturbation
Small Signal Response
Small Signal Gain
Electrical
Sources
Meters
Passive Components
Power Semiconductors

Small Signal Perturbation

Generate a perturbation signal for a small signal analysis. If the feed-through input is shown, the perturbation is added to the input signal.

FIG. 3.199 Small Signal Analysis section of Library Browser.



Block Parameters: BuckConverter/Perturbation

Small Signal Perturbation

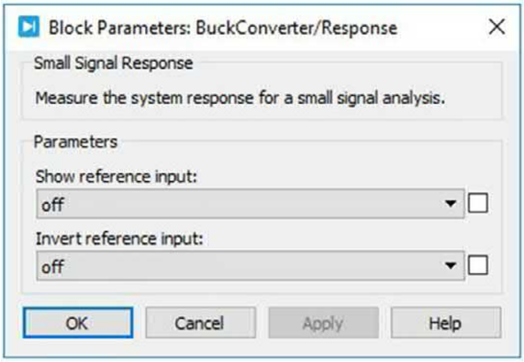
Generate a perturbation signal for a small signal analysis. If the feed-through input is shown, the perturbation is added to the input signal.

Parameters

Show feed-through input: off ☐

OK
Cancel
Apply
Help

FIG. 3.200 "Perturbation" block settings.


Block Parameters: BuckConverter/Response

Small Signal Response

Measure the system response for a small signal analysis.

Parameters

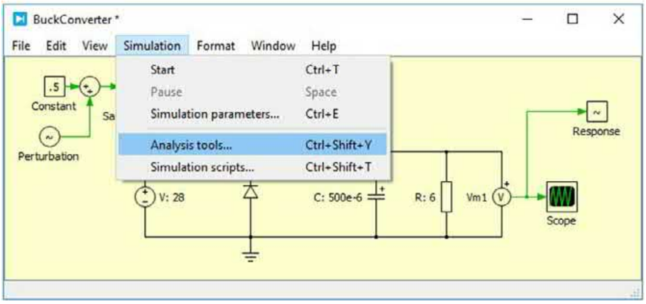
Show reference input: off

Invert reference input: off

☐
☐
OK
Cancel
Apply
Help

FIG. 3.201 “Response” block settings.

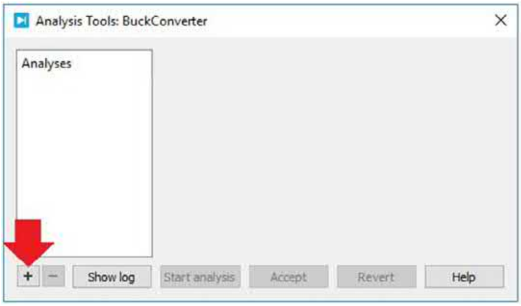
To obtain the small signal transfer function between the duty ratio and output voltage, click on “Analysis tools…” (Fig. 3.202).



BuckConverter *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
.5
Constant
Sa
Perturbation
Response
V: 28
C: 500e-6
R: 6
Vm1
V
Scope

FIG. 3.202 Use “Analysis tools…” to open the Analysis Tools window.

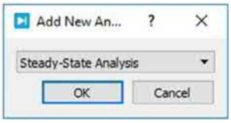
In the opened window, click on the + sign (Fig. 3.203).



Analysis Tools: BuckConverter
Analyses
+
−
Show log
Start analysis
Accept
Revert
Help

FIG. 3.203 Analysis Tools window.

A window such as Fig. 3.204 will appear.


Add New An...
?
OK
Cancel

Steady-State Analysis

FIG. 3.204 Add New Analysis window.

Select the “Impulse Response Analysis” (Fig. 3.205)

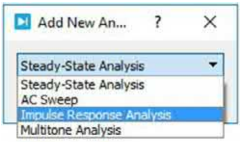
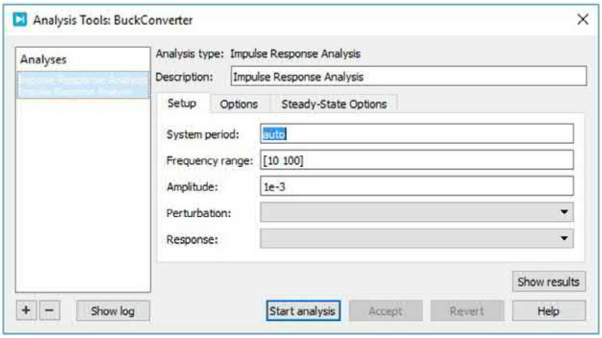

Add New An...
?
Steady-State Analysis
Steady-State Analysis
AC Sweep
Impulse Response Analysis
Multitone Analysis

FIG. 3.205 Select “Impulse Response Analysis” to do small signal analysis.

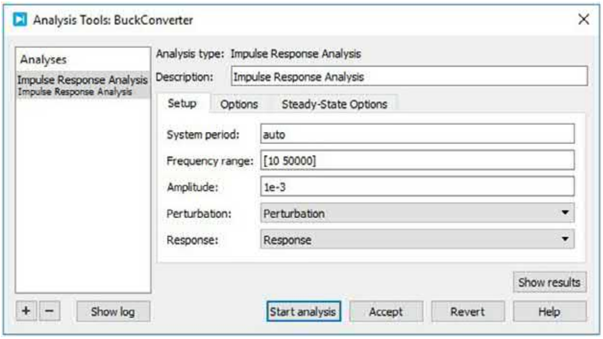
A window such as that shown in Fig. 3.206 will appear.


Analysis Tools: BuckConverter
Analyses
Setup
Options
Steady-State Options
+
−
Show log
Show results
Start analysis
Accept
Revert
Help

Analysis type: Impulse Response Analysis
Description: Impulse Response Analysis
System period: auto
Frequency range: [10 100]
Amplitude: 1e-3
Perturbation:
Response:

FIG. 3.206 Analysis Tools window.

Fill the window like Fig. 3.207. Here we want to calculate the transfer function for 10 Hz-50 KHz range. “Amplitude:” box must contain a small value. The converter steady-state duty ratio is 0.5 (see Fig. 3.198). As a rule of thumb, perturbation amplitude must be less than 0.1 of the average value (in this case, $\frac{0.5}{10} = 0.05$).


Analysis Tools: BuckConverter
Analyses
Impulse Response Analysis
Impulse Response Analysis
Setup
Options
Steady-State Options
+
−
Show log
Show results
Start analysis
Accept
Revert
Help

Analysis type: Impulse Response Analysis
Description: Impulse Response Analysis
System period: auto
Frequency range: [10 50000]
Amplitude: 1e-3
Perturbation: Perturbation
Response: Response

FIG. 3.207 Analysis Tools window after doing the settings.

Click the “Start analysis” button to obtain the small signal Bode plot (Fig. 3.208).

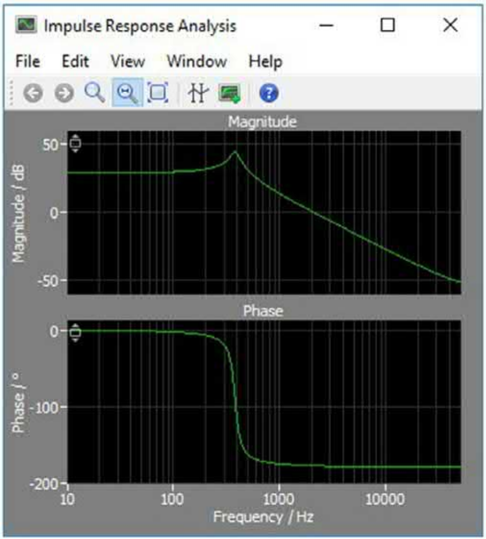
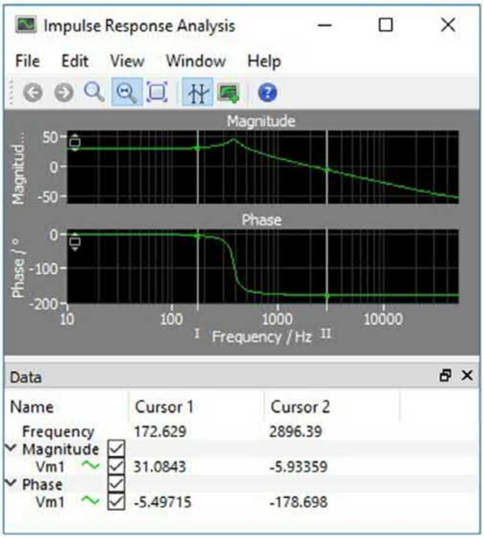

Impulse Response Analysis
File
Edit
View
Window
Help
Magnitude
50
0
-50
Magnitude / dB
Phase
0
-100
-200
Phase / °
10
100
1000
10000
Frequency / Hz

FIG. 3.208 Simulation results $\left(\frac{\tilde{v}_o(s)}{\tilde{d}(s)}\right)$.

You can read the obtained plot with the aid of cursors like other graphs (Fig. 3.209).


Impulse Response Analysis
File
Edit
View
Window
Help
Magnitude
50
0
-50
Magnitud...
Phase
0
-100
-200
Phase / °
10
100
1000
10000
I
Frequency / Hz
II
Data

Name		Cursor 1	Cursor 2
Frequency		172.629	2896.39
Magnitude	☑		
Vm1	☑	31.0843	-5.93359
Phase	☑		
Vm1	☑	-5.49715	-178.698

FIG. 3.209 Adding cursors to the simulation results.

3.7.2 Comparison of different simulation results

Assume that you want to see what happens to the Bode plot if load increases to 12 Ω. Click the "Save trace data…" to save the result of analysis for 6 Ω load. Save the graphics at the desired path (Fig. 3.210).

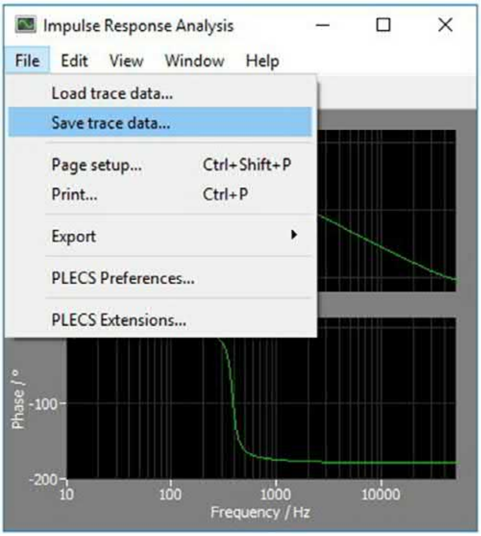
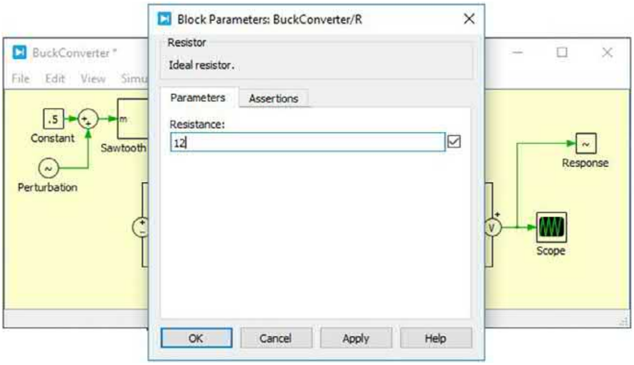

Impulse Response Analysis
File
Edit
View
Window
Help
Load trace data...
Save trace data...
Page setup... Ctrl+Shift+P
Print... Ctrl+P
Export
PLECS Preferences...
PLECS Extensions...
Phase / °
-100
-200
10
100
1000
10000
Frequency / Hz

FIG. 3.210 Use the "Save trace data..." to save the current graph.

After saving the result of analysis for 6-Ω load, return to the schematic and change the load to 12 Ω (Figs. 3.211 and 3.212).


BuckConverter *
File
Edit
View
Simu
.5
Constant
Perturbation
Sawtooth
m
Response
Scope
V
Block Parameters: BuckConverter/R
Parameters
Assertions
OK
Cancel
Apply
Help

Resistor

Ideal resistor.

Resistance: 12 ☑

FIG. 3.211 Increasing the load resistance to 12 Ω.

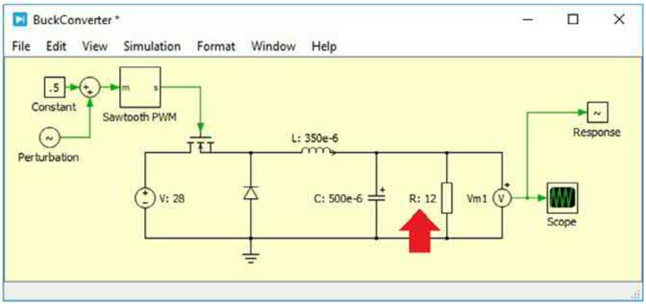

BuckConverter *
File
Edit
View
Simulation
Format
Window
Help
.5
Constant
Perturbation
Sawtooth PWM
m
s
L: 350e-6
V: 28
C: 500e-6
R: 12
Vm1
V
Response
Scope

FIG. 3.212 Schematic after increasing the load resistance.

Go to the "Analysis tools…" (Fig. 3.213).

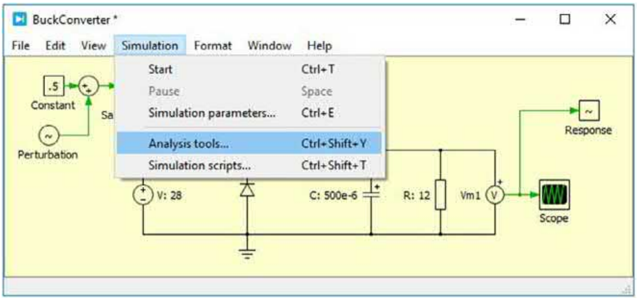
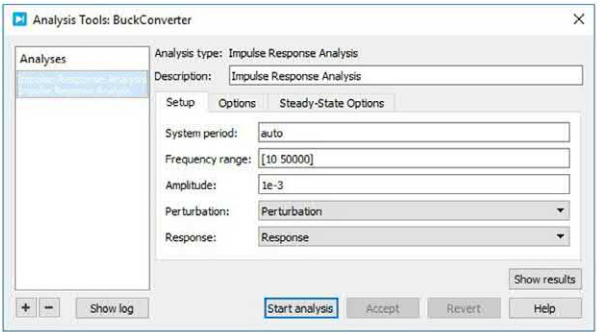

BuckConverter *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
.5
Constant
Perturbation
Sa
V: 28
C: 500e-6
R: 12
Vm1
V
Response
Scope

FIG. 3.213 Use "Analysis tools…" to open the Analysis Tools window.

Click the "Start analysis" button to obtain the result for 12-Ω load (Fig. 3.214).


Analysis Tools: BuckConverter
Analyses
Setup
Options
Steady-State Options
Show results
+
−
Show log
Start analysis
Accept
Revert
Help

Analysis type: Impulse Response Analysis
Description: Impulse Response Analysis
System period: auto
Frequency range: [10 50000]
Amplitude: 1e-3
Perturbation: Perturbation
Response: Response

FIG. 3.214 Analysis Tools window.

Fig. 3.215 shows the analysis result.

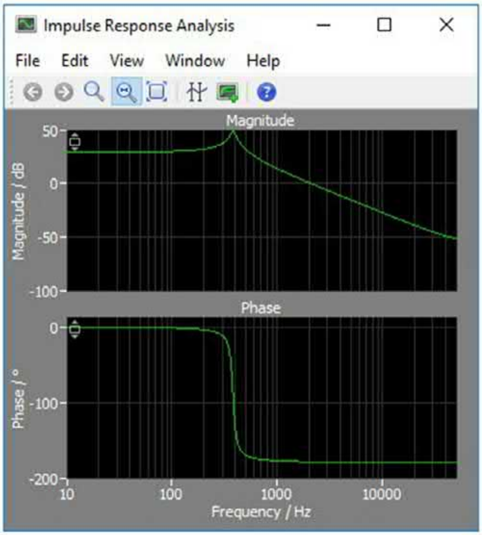

Impulse Response Analysis
File
Edit
View
Window
Help
Magnitude
Magnitude / dB
50
0
-50
-100
Phase
Phase / °
0
-100
-200
10
100
1000
10000
Frequency / Hz

FIG. 3.215 Simulation results $\left(\frac{\tilde{v}_o(s)}{\tilde{d}(s)}\right)$ for R = 12 Ω.

Click the "Load trace data…" and read the file of 6-Ω analysis (Fig. 3.216).

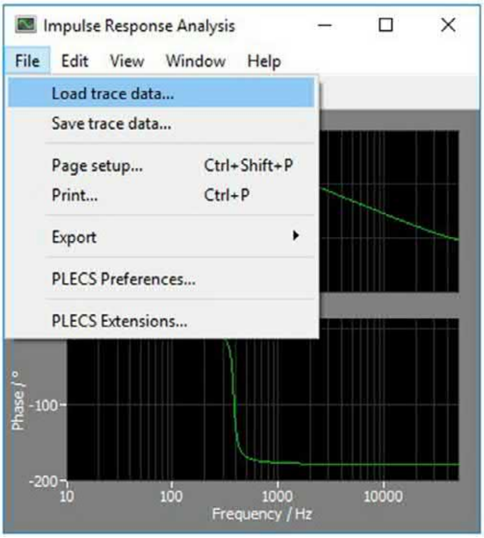

Impulse Response Analysis
File
Edit
View
Window
Help
Load trace data...
Save trace data...
Page setup... Ctrl+Shift+P
Print... Ctrl+P
Export
PLECS Preferences...
PLECS Extensions...
Phase / °
-100
-200
10
100
1000
10000
Frequency / Hz

FIG. 3.216 Use "Load trace data…" to load the saved analysis result (for R = 6 Ω).

PLECS shows both of the results on the same graph. This makes comparison possible (Fig. 3.217).

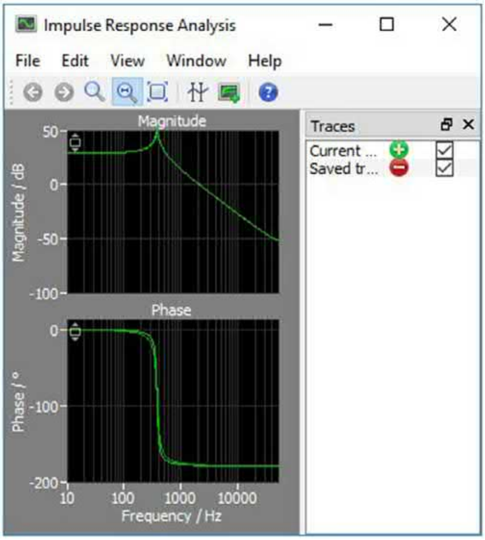

Impulse Response Analysis
File Edit View Window Help
Magnitude
Magnitude / dB
50
0
-50
-100
Phase
Phase / °
0
-100
-200
10
100
1000
10000
Frequency / Hz
Traces
Current ...
Saved tr...

FIG. 3.217 Traces section is added to the window.

Use magnifier to get a better view. You can close the "Traces" window by clicking on the × icon (Fig. 3.218).

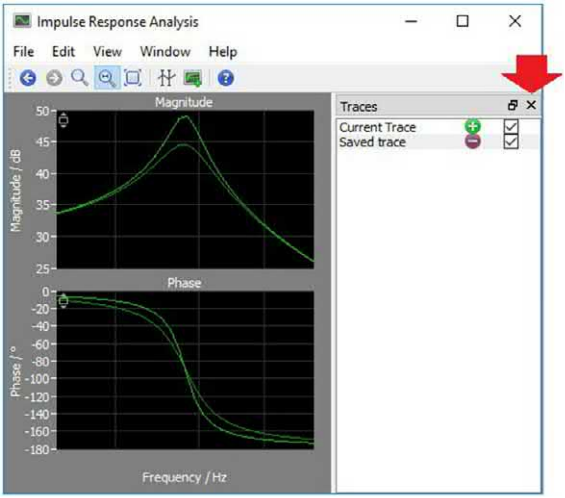

Impulse Response Analysis
File Edit View Window Help
Magnitude
Magnitude / dB
50
45
40
35
30
25
Phase
Phase / °
0
-20
-40
-60
-80
-100
-120
-140
-160
-180
Frequency / Hz
Traces
Current Trace
Saved trace

FIG. 3.218 Zooming in the results.

You can change the graphs color by right clicking on the graph and select the “Edit curve properties” (Fig. 3.219)

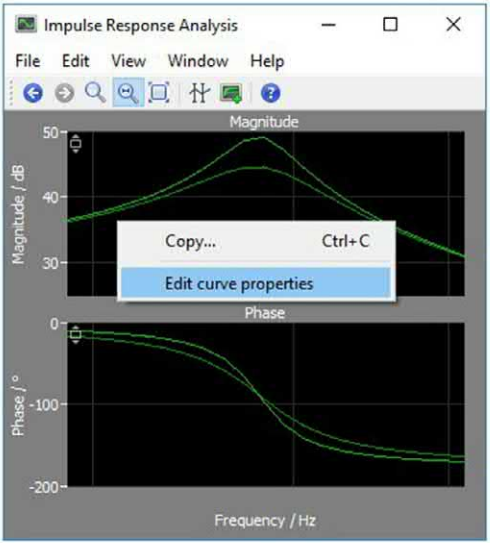
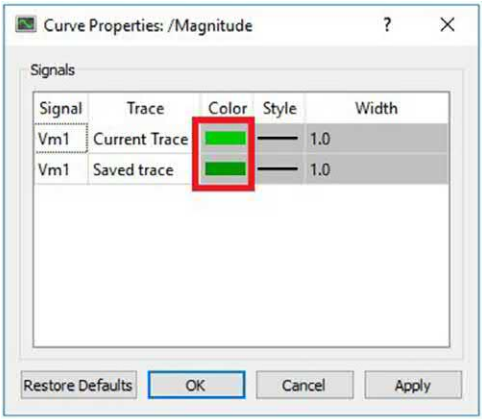

Impulse Response Analysis
File
Edit
View
Window
Help
Magnitude
50
40
30
Magnitude / dB
Copy...
Ctrl+C
Edit curve properties
Phase
0
-100
-200
Phase / °
Frequency / Hz

FIG. 3.219 Right click on the graph and select the “Edit curve properties” to change the curves colors, style, and line width.

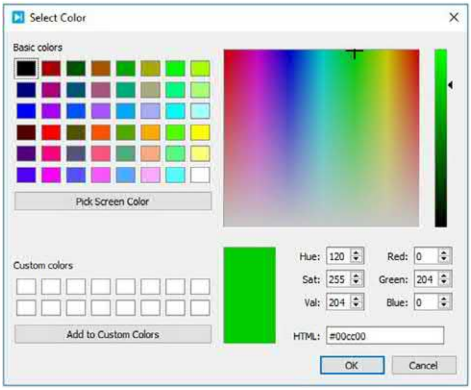
Click on “Color” section to select the desired colors (Fig. 3.220).


Curve Properties: /Magnitude
?
Signals
Restore Defaults
OK
Cancel
Apply

Signal	Trace	Color	Style	Width
Vm1	Current Trace	🟩	—	1.0
Vm1	Saved trace	🟩	—	1.0

FIG. 3.220 Curve Properties window.

Select your favorite color from the opened window (Fig. 3.221).


Select Color
Basic colors
Pick Screen Color
Custom colors
Add to Custom Colors
OK
Cancel

Hue: 120
Sat: 255
Val: 204
Red: 0
Green: 204
Blue: 0
HTML: #00cc00

FIG. 3.221 Select Color window.

You can obtain other small signal transfer functions in the same way you obtain the control to output voltage transfer function. For instance, if you want to obtain the control to inductor current transfer function, change the schematic into the one shown in Fig. 3.222.

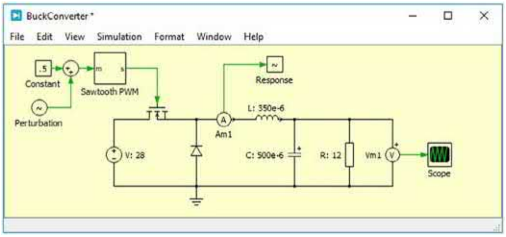

BuckConverter *
File
Edit
View
Simulation
Format
Window
Help
.5
Constant
m
s
Sawtooth PWM
Perturbation
Response
L: 350e-6
A
Am1
V: 28
C: 500e-6
R: 12
Vm1
V
Scope

FIG. 3.222 Schematic to obtain the control to inductor current transfer function $\left(\frac{\tilde{i}_L(s)}{\tilde{d}(s)}\right)$.

Analysis result for Fig. 3.222 is shown in Fig. 3.223.

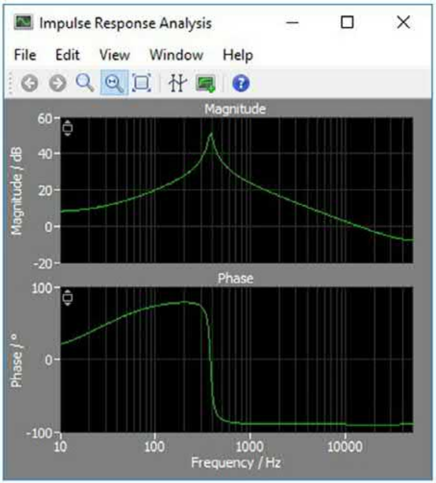


FIG. 3.223 Simulation result $\left(\frac{\tilde{i}_L(s)}{\tilde{d}(s)}\right)$.

3.7.3 Importing the simulation results into the MATLAB

You can transfer the obtained Bode plot into the MATLAB for further process. To do this, you must export the obtained result as a CSV file. To do this click on "All…" (Fig. 3.224).

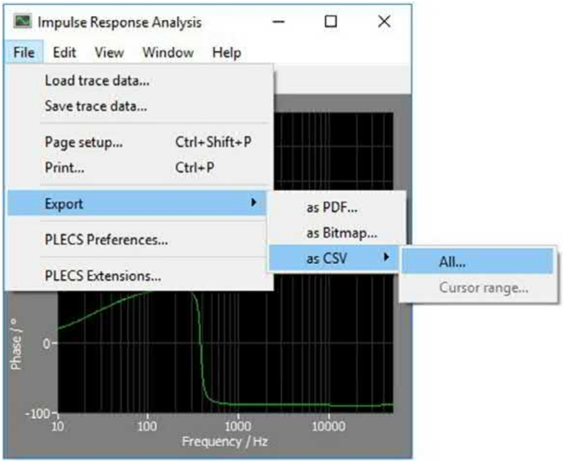
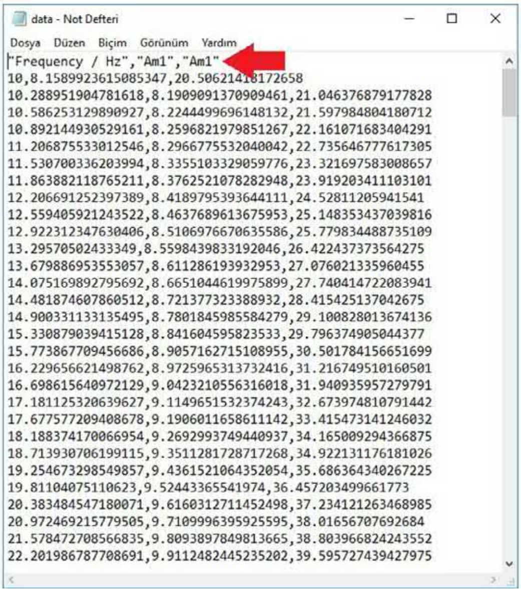

Impulse Response Analysis
File
Edit
View
Window
Help
Load trace data...
Save trace data...
Page setup... Ctrl+Shift+P
Print... Ctrl+P
Export
PLECS Preferences...
PLECS Extensions...
as PDF...
as Bitmap...
as CSV
All...
Cursor range...
Phase / °
0
-100
10
100
1000
10000
Frequency / Hz

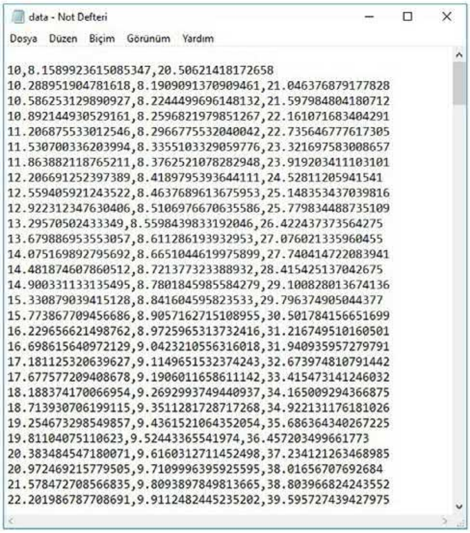
FIG. 3.224 Exporting the graph as .csv file.

Save the CSV file to desired path. Here we saved it in the "C:\" path and named it data.csv. After saving, open the file in the Notepad and remove the first line. After removing the first line, save the file (Figs. 3.225 and 3.226).


data - Not Defteri
Dosya
Düzen
Biçim
Görünüm
Yardım

```
"Frequency / Hz","Am1","Am1"
10,8.1589923615085347,20.50621418172658
10.288951904781618,8.1909091370909461,21.046376879177828
10.586253129890927,8.2244499696148132,21.597984804180712
10.892144930529161,8.2596821979851267,22.161071683404291
11.206875533012546,8.2966775532040042,22.735646777617305
11.530700336203994,8.3355103329059776,23.321697583008657
11.863882118765211,8.3762521078282948,23.919203411103101
12.206691252397389,8.4189795393644111,24.52811205941541
12.559405921243522,8.4637689613675953,25.148353437039816
12.922312347630406,8.5106976670635586,25.779834488735109
13.29570502433349,8.5598439833192046,26.422437373564275
13.679886953553057,8.611286193932953,27.07602133596045
14.075169892795692,8.6651044619975899,27.740414722083941
14.481874607860512,8.721377323388932,28.415425137042675
14.900331133135495,8.7801845985584279,29.100828013674136
15.330879039415128,8.841604595823533,29.796374905044377
15.773867709456686,8.9057162715108955,30.501784156651699
16.229656621498762,8.9725965313732416,31.216749510160501
16.698615640972129,9.0423210556316018,31.940935957279791
17.181125320639627,9.1149651532374243,32.673974810791442
17.677577209408678,9.1906011658611142,33.415473141246032
18.188374170066954,9.2692993749440937,34.165009294366875
18.713930706199115,9.3511281728717268,34.922131176181026
19.254673298549857,9.4361521064352054,35.686364340267225
19.81104075110623,9.52443365541974,36.457203499661773
20.383484547180071,9.6160312711452498,37.234121263468985
20.972469215779505,9.7109996395925595,38.01656707692684
21.578472708566835,9.8093897849813665,38.803966824243552
22.201986787708691,9.9112482445235202,39.595727439427975
```

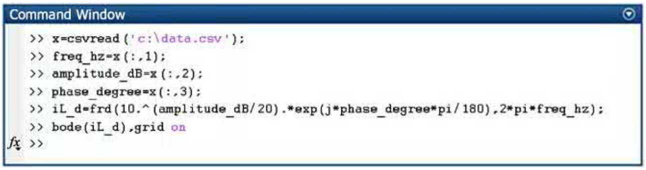
FIG. 3.225 Opening the data.csv file in Notepad and removing the first line.



```
10,8.1589923615085347,20.50621418172658
10.288951904781618,8.1909091370909461,21.046376879177828
10.586253129890927,8.2244499696148132,21.597984804180712
10.892144930529161,8.2596821979851267,22.161071683404291
11.206875533012546,8.2966775532040042,22.735646777617305
11.530700336203994,8.3355103329059776,23.321697583008657
11.863882118765211,8.3762521078282948,23.919203411103101
12.206691252397389,8.4189795393644111,24.52811205941541
12.559405921243522,8.4637689613675953,25.148353437039816
12.922312347630406,8.5106976670635586,25.779834488735109
13.29570502433349,8.5598439833192046,26.422437373564275
13.679886953553057,8.611286193932953,27.076021335960455
14.075169892795692,8.6651044619975899,27.740414722083941
14.481874607860512,8.721377323388932,28.415425137042675
14.900331133135495,8.7801845985584279,29.100828013674136
15.330879039415128,8.841604595823533,29.796374905044377
15.773867709456686,8.9057162715108955,30.501784156651699
16.229656621498762,8.9725965313732416,31.216749510160501
16.698615640972129,9.0423210556316018,31.940935957279791
17.181125320639627,9.1149651532374243,32.673974810791442
17.677577209408678,9.1906011658611142,33.415473141246032
18.188374170066954,9.2692993749440937,34.165009294366875
18.713930706199115,9.3511281728717268,34.922131176181026
19.254673298549857,9.4361521064352054,35.686364340267225
19.81104075110623,9.52443365541974,36.457203499661773
20.383484547180071,9.6160312711452498,37.234121263468985
20.972469215779505,9.7109996395925595,38.01656707692684
21.578472708566835,9.8093897849813665,38.803966824243552
22.201986787708691,9.9112482445235202,39.595727439427975
```

FIG. 3.226 Removing the first line.

Run the MATLAB. Write the codes shown in Fig. 3.227. This imports the data into the MATLAB (Fig. 3.228).



```
>> x=csvread('c:\data.csv');
>> freq_hz=x(:,1);
>> amplitude_dB=x(:,2);
>> phase_degree=x(:,3);
>> iL_d=frd(10.^(amplitude_dB/20).*exp(j*phase_degree*pi/180),2*pi*freq_hz);
>> bode(iL_d),grid on
fx >>
```

FIG. 3.227 Reading the data.csv in MATLAB.

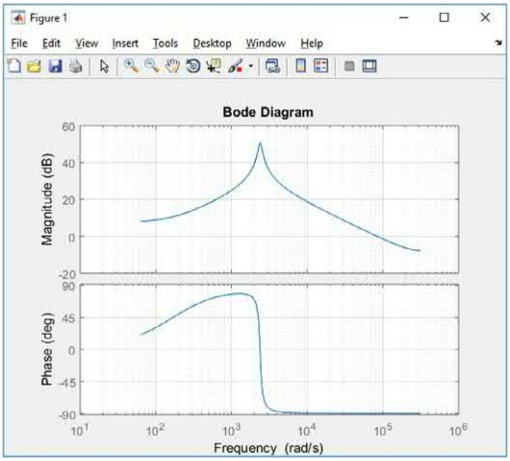


FIG. 3.228 PLECS graph is regenerated in MATLAB (see Fig. 3.223).

3.8 Example 3.8: Mutual inductance

In this example, we will show how you can simulate circuits containing mutual inductors.

3.8.1 Preparing the simulation

Assume you want to analyze the circuit shown in Fig. 3.229.

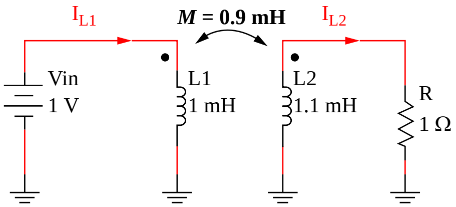


FIG. 3.229 A simple circuit containing the mutual inductors.

According to basic circuit theory (initial condition assumed to be zero),

$$\begin{cases} L1\dfrac{di_{L1}}{dt}-M\dfrac{di_{L2}}{dt}=V_{in}(\mathrm{t}) \\ Ri_{L2}+L_2\dfrac{di_{L2}}{dt}-M\dfrac{di_{L1}}{dt}=0 \end{cases}$$

$$\begin{cases} L1\dfrac{di_{L1}}{dt}-M\dfrac{di_{L2}}{dt}=V_{in}(\mathrm{t}) \\ Ri_{L2}+L_2\dfrac{di_{L2}}{dt}-M\dfrac{di_{L1}}{dt}=0 \end{cases}$$

If we take the Laplace transform of both sides,

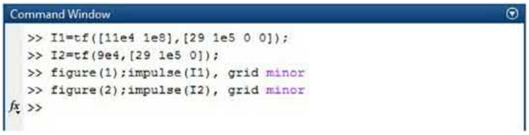
$$\begin{bmatrix} L_1s & -Ms \\ -Ms & R+L_2s \end{bmatrix}\times\begin{bmatrix} I_{L1}(s) \\ I_{L2}(s) \end{bmatrix}=\begin{bmatrix} V_{in}(s) \\ 0 \end{bmatrix}$$

$$\begin{bmatrix} I_{L1}(s) \\ I_{L2}(s) \end{bmatrix}=\begin{bmatrix} L_1s & -Ms \\ -Ms & R+L_2s \end{bmatrix}^{-1}\times\begin{bmatrix} V_{in}(s) \\ 0 \end{bmatrix}$$

is obtained. Since Vin(t)=1 V, its Laplace transform is Vin(s) $=\frac{1}{s}$. So, Laplace transform of currents is obtained as:

$$\begin{bmatrix} I_{L1}(s) \\ I_{L2}(s) \end{bmatrix}=\begin{bmatrix} \dfrac{(11s+10000)\times 10000}{s^2\times(29s+100000)} \\ \dfrac{90000}{s(29s+100000)} \end{bmatrix}$$

We can draw the time domain graph of these equations using commands shown in Fig. 3.230. Results are shown in Figs. 3.231 and 3.232.



```
>> I1=tf([11e4 1e8],[29 1e5 0 0]);
>> I2=tf(9e4,[29 1e5 0]);
>> figure(1);impulse(I1), grid minor
>> figure(2);impulse(I2), grid minor
fx >>
```

FIG. 3.230 MATLAB codes to obtain the time domain graph of $I_{L1}(s)$ and $I_{L2}(s)$.

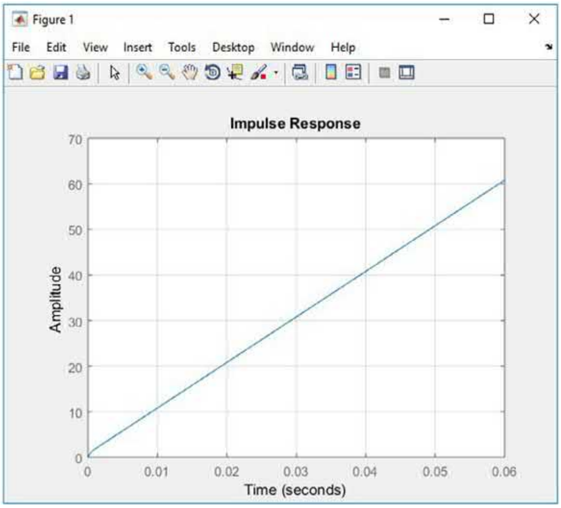


FIG. 3.231 Time domain graph of $I_{L1}(s)$.

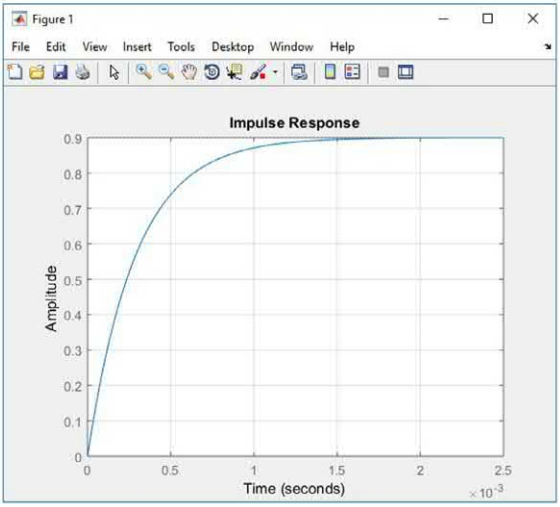


FIG. 3.232 Time domain graph of $I_{L2}(s)$.

In order to analyze the circuit using PLECS, we draw the schematic shown in Fig. 3.233 This schematic uses the mutual inductor block, which can be found under “Passive Components” section of Library Browser (see Fig. 3.234). As shown in Fig. 3.235, the first winding is shown with a small circle. Other windings are shown with dot.

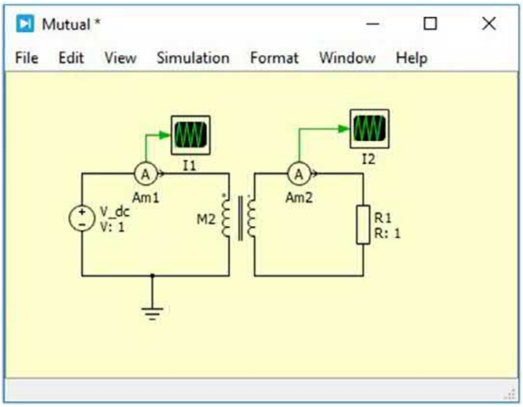

Mutual *
File
Edit
View
Simulation
Format
Window
Help
I1
I2
A
Am1
A
Am2
V_dc
V: 1
M2
R1
R: 1

FIG. 3.233 Schematic to simulate the circuit shown in Fig. 3.229.

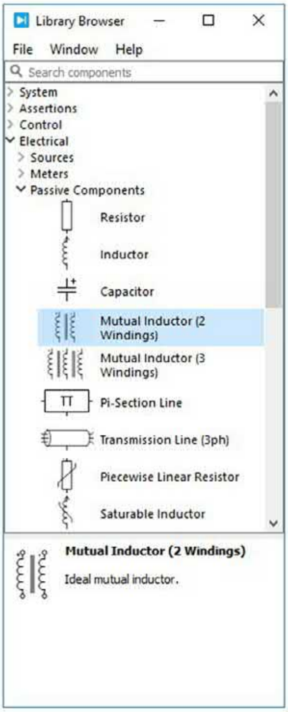

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Resistor
Inductor
Capacitor
Mutual Inductor (2 Windings)
Mutual Inductor (3 Windings)
TT
Pi-Section Line
Transmission Line (3ph)
Piecewise Linear Resistor
Saturable Inductor
Mutual Inductor (2 Windings)
Ideal mutual inductor.

FIG. 3.234 Mutual Inductor block.

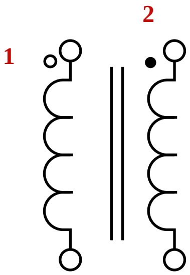
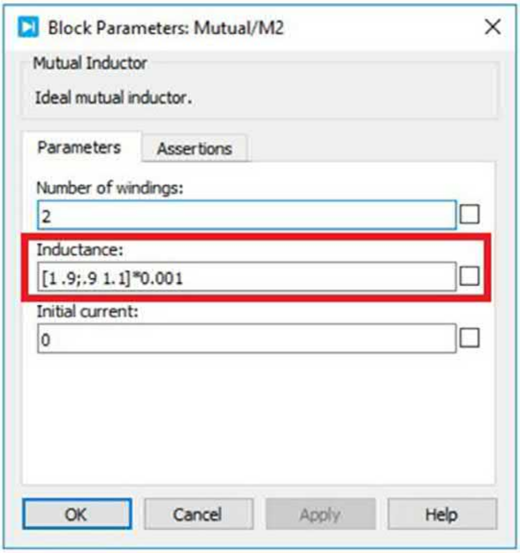


FIG. 3.235 First winding is shown with a small circuit. Other windings are shown with a small dot.

Double click on the mutual inductor block and do the setting as shown in Fig. 3.236. With these settings the mutual inductor simulate the mutual inductor block shown in Fig. 3.229. The "Inductance:" box takes a matrix. The main diagonal shows the self-inductance of windings. Other elements of the matrix show the mutual inductance between windings (Fig. 3.236). If you right click on the mutual inductance block and select the Help you can see more information about the block (Fig. 3.237).


Block Parameters: Mutual/M2

Mutual Inductor

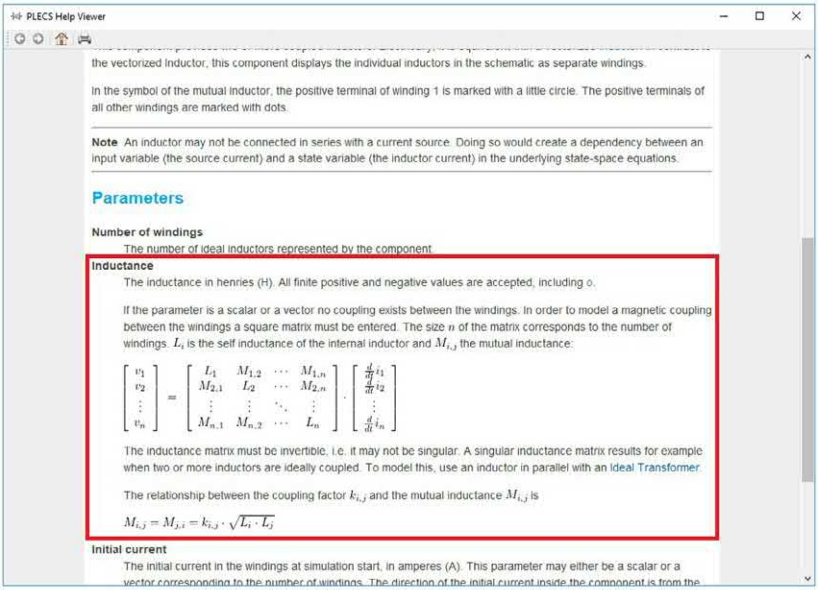
Ideal mutual inductor.

Number of windings: 2 ☐

Inductance: [1 .9;.9 1.1]*0.001 ☐

Initial current: 0 ☐

FIG. 3.236 Setting the parameters of the mutual inductor.



the vectorized Inductor, this component displays the individual inductors in the schematic as separate windings.

In the symbol of the mutual inductor, the positive terminal of winding 1 is marked with a little circle. The positive terminals of all other windings are marked with dots.

Note An inductor may not be connected in series with a current source. Doing so would create a dependency between an input variable (the source current) and a state variable (the inductor current) in the underlying state-space equations.

Parameters

Number of windings

The number of ideal inductors represented by the component.

Inductance

The inductance in henries (H). All finite positive and negative values are accepted, including 0.

If the parameter is a scalar or a vector no coupling exists between the windings. In order to model a magnetic coupling between the windings a square matrix must be entered. The size n of the matrix corresponds to the number of windings. L_i is the self inductance of the internal inductor and $M_{i,j}$ the mutual inductance:

$$\begin{bmatrix} v_1 \\ v_2 \\ \vdots \\ v_n \end{bmatrix} = \begin{bmatrix} L_1 & M_{1,2} & \cdots & M_{1,n} \\ M_{2,1} & L_2 & \cdots & M_{2,n} \\ \vdots & \vdots & \ddots & \vdots \\ M_{n,1} & M_{n,2} & \cdots & L_n \end{bmatrix} \cdot \begin{bmatrix} \frac{d}{dt} i_1 \\ \frac{d}{dt} i_2 \\ \vdots \\ \frac{d}{dt} i_n \end{bmatrix}$$

The inductance matrix must be invertible, i.e. it may not be singular. A singular inductance matrix results for example when two or more inductors are ideally coupled. To model this, use an inductor in parallel with an Ideal Transformer.

The relationship between the coupling factor $k_{i,j}$ and the mutual inductance $M_{i,j}$ is

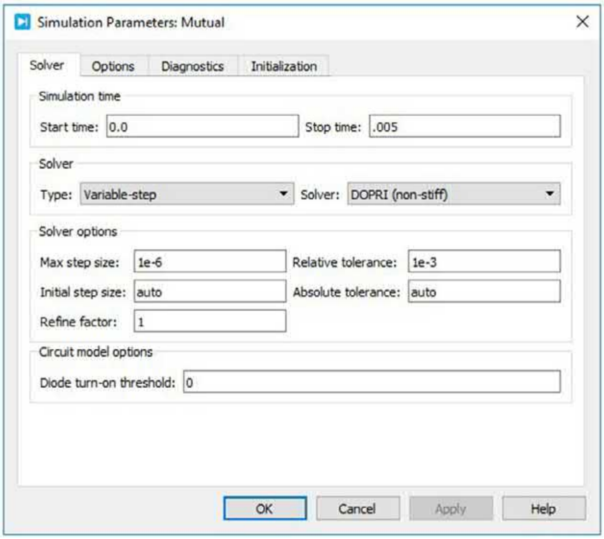
$$M_{i,j} = M_{j,i} = k_{i,j} \cdot \sqrt{L_i \cdot L_j}$$

Initial current

The initial current in the windings at simulation start, in amperes (A). This parameter may either be a scalar or a vector corresponding to the number of windings. The direction of the initial current inside the component is from the

FIG. 3.237 Documentation for mutual inductor block.

If we run the simulation with the setting shown in Fig. 3.238, the results shown in Figs. 3.239 and 3.240 will be obtained. Compare them with Fig. 3.231 and 3.232. The results are the same.



Start time: 0.0

Stop time: .005

Type: Variable-step

Solver: DOPRI (non-stiff)

Max step size: 1e-6

Relative tolerance: 1e-3

Initial step size: auto

Absolute tolerance: auto

Refine factor: 1

Diode turn-on threshold: 0

FIG. 3.238 Simulation parameters.

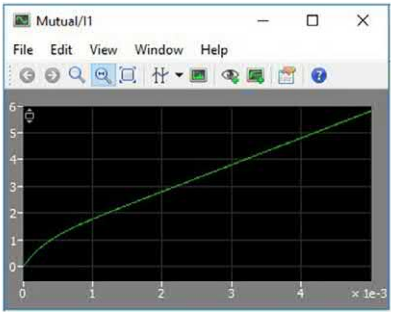


FIG. 3.239 Primary winding current.

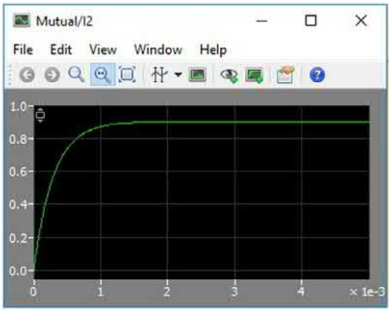


FIG. 3.240 Secondary winding current.

If your schematic needs transformers, you can find it in the Transformer section of Library Browser (Fig. 3.241).

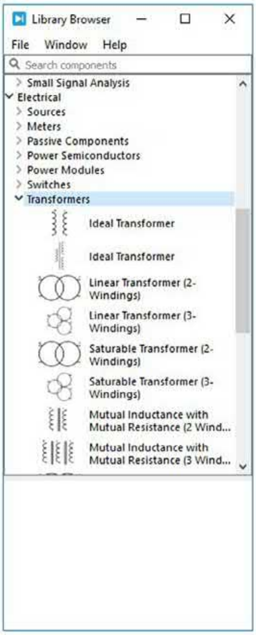
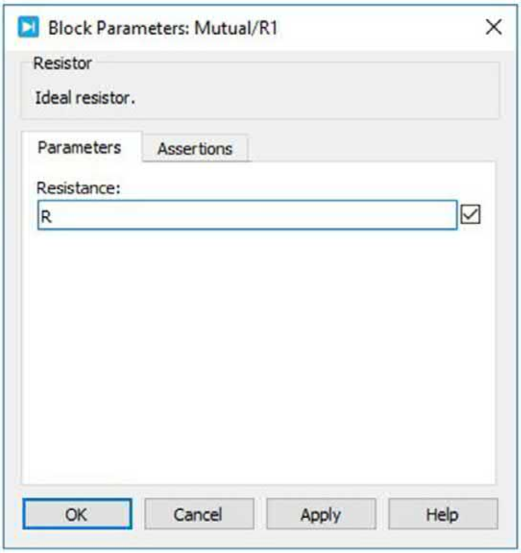

Library Browser
File Window Help
Search components
Small Signal Analysis
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Ideal Transformer
Ideal Transformer
Linear Transformer (2-Windings)
Linear Transformer (3-Windings)
Saturable Transformer (2-Windings)
Saturable Transformer (3-Windings)
Mutual Inductance with Mutual Resistance (2 Wind...
Mutual Inductance with Mutual Resistance (3 Wind...

FIG. 3.241 Transformers section of Library Browser.

3.8.2 Using parametric variables to specify the component values

Up to now, we used the numeric values to fill the components settings. PLECS allows you to use variables as well. The process of defining variables is shown with the aid of an example.

Double click on the resistor block and change the “Resistance:” box to “R” (Figs. 3.242 and 3.243).


Block Parameters: Mutual/R1
Resistor
Ideal resistor.
Parameters
Assertions
OK
Cancel
Apply
Help

Resistance: R ☑

FIG. 3.242 The Resistance box is not filled with a numeric value.

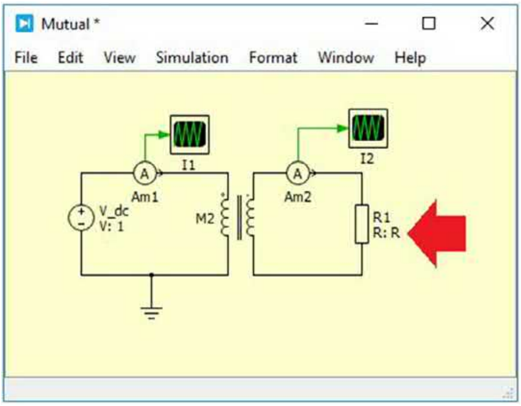

Mutual *
File
Edit
View
Simulation
Format
Window
Help
I1
I2
A
Am1
A
Am2
V_dc
V: 1
M2
R1
R: R

FIG. 3.243 The schematic after changing the R1 resistor value to R.

Click the “Simulation parameters…” (Fig. 3.244).

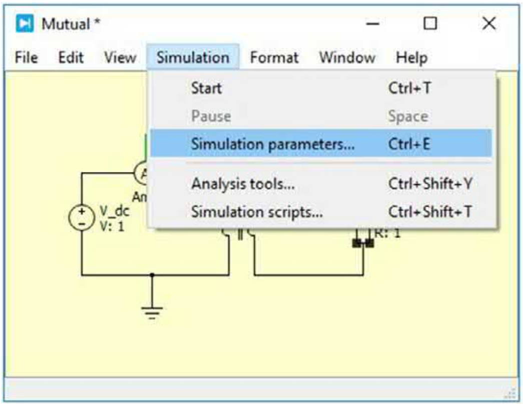
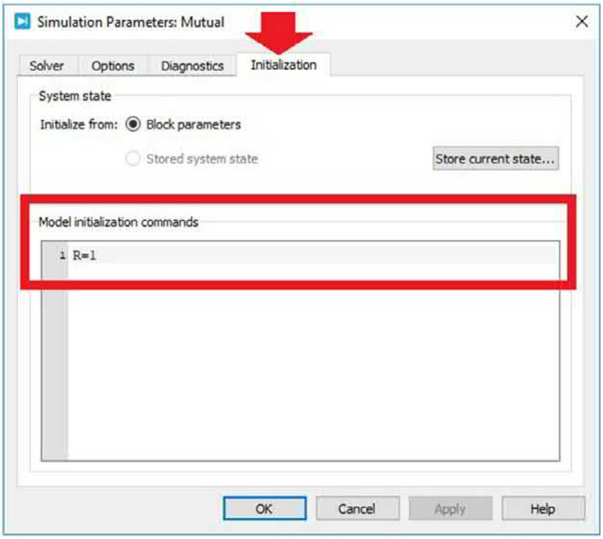

Mutual *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
V_dc
V: 1
Am
R: 1

FIG. 3.244 Opening the Simulation Parameters window by clicking the “Simulation parameters…”.

PLECS opens the “Simulation Parameters” window. Go to the “Initialization” pane. Write R = 1 in the “Model initialization commands” section (Fig. 3.245).


Simulation Parameters: Mutual
Solver
Options
Diagnostics
Initialization
System state
Store current state...
OK
Cancel
Apply
Help

Initialize from: ● Block parameters

○ Stored system state

Model initialization commands

```
R=1
```

FIG. 3.245 Initialization tab of Simulation Parameters window.

Using the aforementioned method, you can initialize the schematic more rapidly and more easily. Assume you have a symmetric 3-phase RL load. If you do not use the initialization, you must set 6 parameters if you decide to see the analysis results as load changes. Using initialization, you change just two parameters for each simulation.

3.9 Example 3.9: 3-Phase inverter

In this example, we will simulate a 3-phase inverter using the Sine Pulse Width Modulation (SVPWM) technique.

3.9.1 Preparing the simulation

The schematic of a 3-phase inverter is shown in Fig. 3.246. The Schematic is composed of three parts:

- PWM Generator
- Power Stage
- Load

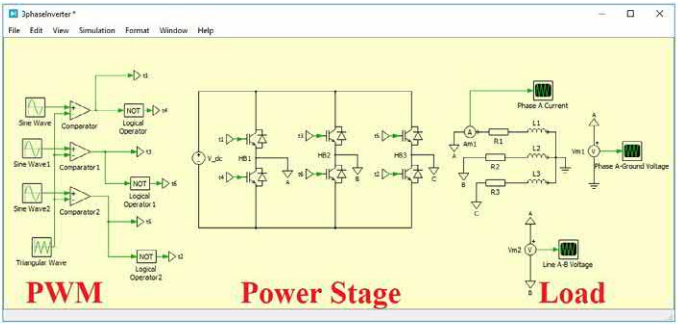

3phaseInverter *
File
Edit
View
Simulation
Format
Window
Help
t1
Sine Wave
Comparator
NOT
Logical Operator
t4
Sine Wave1
Comparator1
t3
NOT
Logical Operator 1
t6
Sine Wave2
Comparator2
t5
Triangular Wave
NOT
Logical Operator2
t2
V_dc
HB1
HB2
HB3
A
B
C
Phase A Current
Am1
R1
L1
R2
L2
R3
L3
Vm1
Phase A-Ground Voltage
Vm2
Line A-B Voltage
PWM
Power Stage
Load

FIG. 3.246 Simulation diagram of 3-phase inverter.

3.9.1.1 *PWM Generator*

The PWM generator section of the inverter is shown in Fig. 3.247. It uses Sine PWM (SPWM) to generate control signals. The PWM generator is composed of “Comparator,” “Logical Operator,” and “Sine Wave Generator” blocks. See Figs. 3.248–3.250 to learn their places.

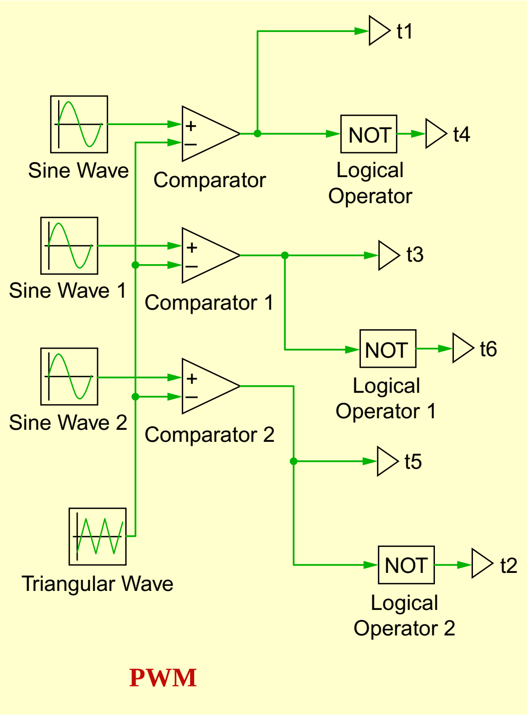
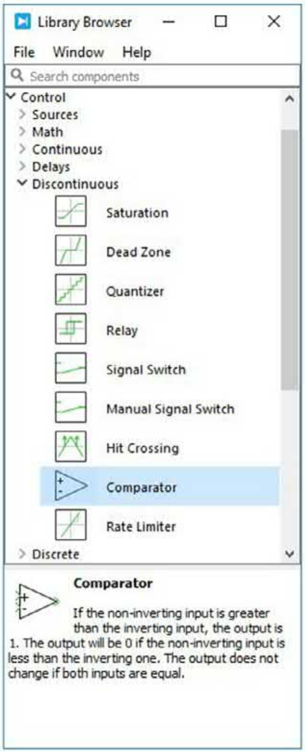

t1
Sine Wave
Comparator
NOT
Logical Operator
t4
Sine Wave 1
Comparator 1
t3
NOT
Logical Operator 1
t6
Sine Wave 2
Comparator 2
t5
Triangular Wave
NOT
Logical Operator 2
t2
PWM

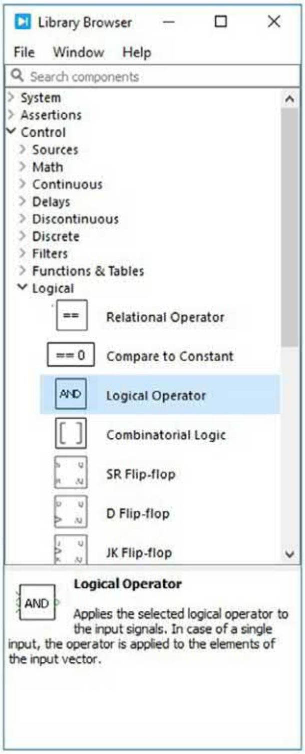
FIG. 3.247 PWM generation section.


Library Browser
File Window Help
Search components
Control
Sources
Math
Continuous
Delays
Discontinuous
Saturation
Dead Zone
Quantizer
Relay
Signal Switch
Manual Signal Switch
Hit Crossing
Comparator
Rate Limiter
Discrete

Comparator

If the non-inverting input is greater than the inverting input, the output is 1. The output will be 0 if the non-inverting input is less than the inverting one. The output does not change if both inputs are equal.

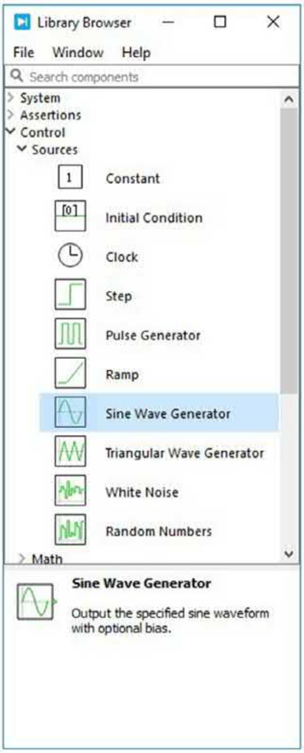
FIG. 3.248 Comparator block.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
==
Relational Operator
== 0
Compare to Constant
AND
Logical Operator
[ ]
Combinatorial Logic
SR Flip-flop
D Flip-flop
JK Flip-flop
AND

Logical Operator

Applies the selected logical operator to the input signals. In case of a single input, the operator is applied to the elements of the input vector.

FIG. 3.249 Logical Operator block.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
1
Constant
[0]
Initial Condition
Clock
Step
Pulse Generator
Ramp
Sine Wave Generator
Triangular Wave Generator
White Noise
Random Numbers
Math

Sine Wave Generator

Output the specified sine waveform with optional bias.

FIG. 3.250 Sine Wave Generator block.

When you place the "Logical Operator" block on the schematic, it will be shown as Fig. 3.251. It must be converted into a NOT gate.

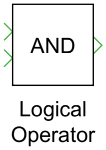
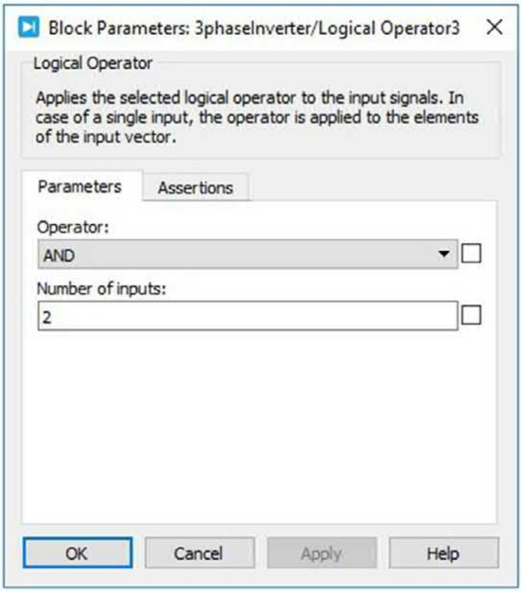

AND
Logical Operator

FIG. 3.251 Logical Operator block.

Double click on the Logical Operator block (Fig. 3.252).


Block Parameters: 3phaseInverter/Logical Operator3

Logical Operator

Applies the selected logical operator to the input signals. In case of a single input, the operator is applied to the elements of the input vector.

Parameters
Assertions

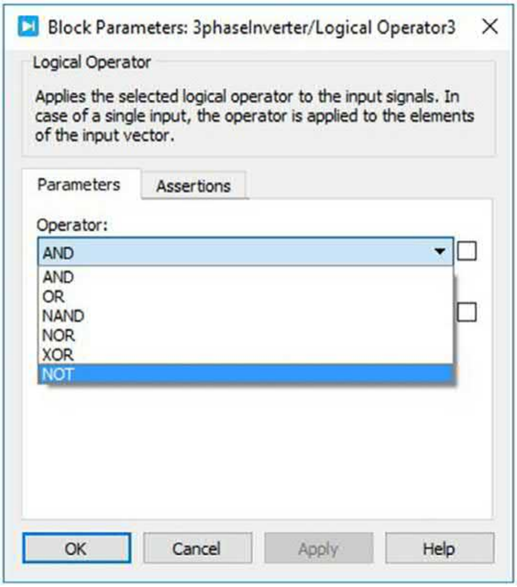
Operator: AND ☐

Number of inputs: 2 ☐

OK
Cancel
Apply
Help

FIG. 3.252 Logical Operator block settings.

Select the "NOT" from the "Operator" drop down list (Fig. 3.253).



Block Parameters: 3phaseInverter/Logical Operator3

Logical Operator

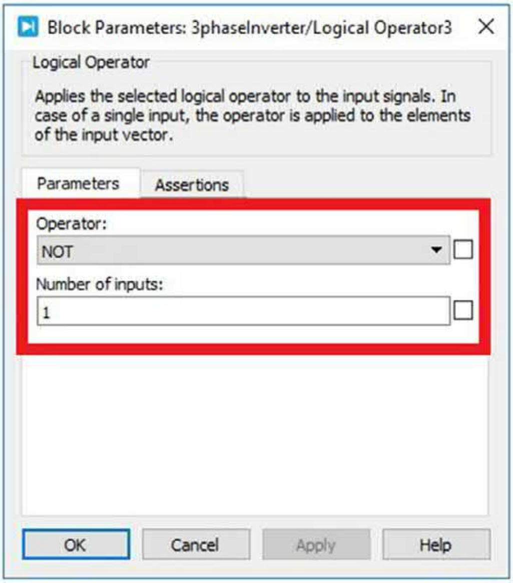
Applies the selected logical operator to the input signals. In case of a single input, the operator is applied to the elements of the input vector.

Operator: AND

Parameters
Assertions
AND
OR
NAND
NOR
XOR
NOT
OK
Cancel
Apply
Help

FIG. 3.253 Turning the logical operator block into a NOT gate.

Set the “Number of inputs:” to 1 (Fig. 3.254).



Block Parameters: 3phaseInverter/Logical Operator3

Logical Operator

Applies the selected logical operator to the input signals. In case of a single input, the operator is applied to the elements of the input vector.

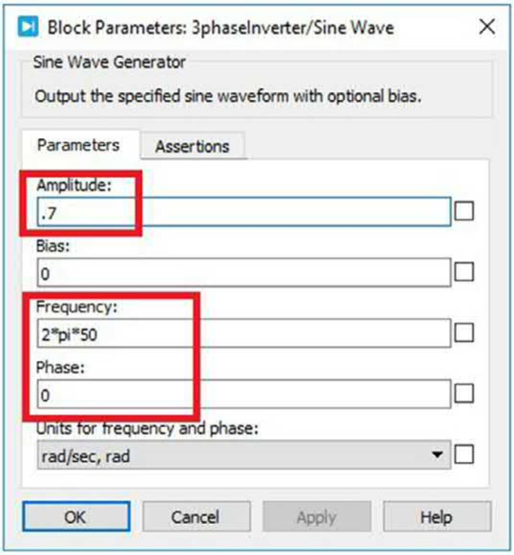
Operator: NOT

Number of inputs: 1

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.254 Turning the Number of inputs into 1.

"Sine Wave Generator" blocks settings are shown in Figs. 3.255–3.257.


Block Parameters: 3phaseInverter/Sine Wave

Sine Wave Generator

Output the specified sine waveform with optional bias.

Amplitude: .7 ☐

Bias: 0 ☐

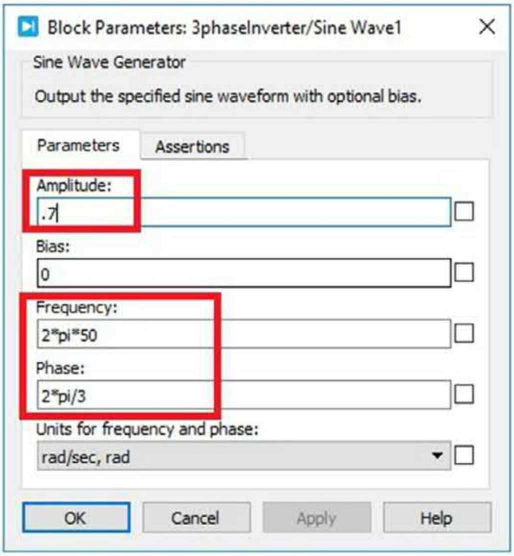
Frequency: 2*pi*50 ☐

Phase: 0 ☐

Units for frequency and phase: rad/sec, rad ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.255 "Sine Wave" block settings (see Fig. 3.247).


Block Parameters: 3phaseInverter/Sine Wave1

Sine Wave Generator

Output the specified sine waveform with optional bias.

Amplitude: .7 ☐

Bias: 0 ☐

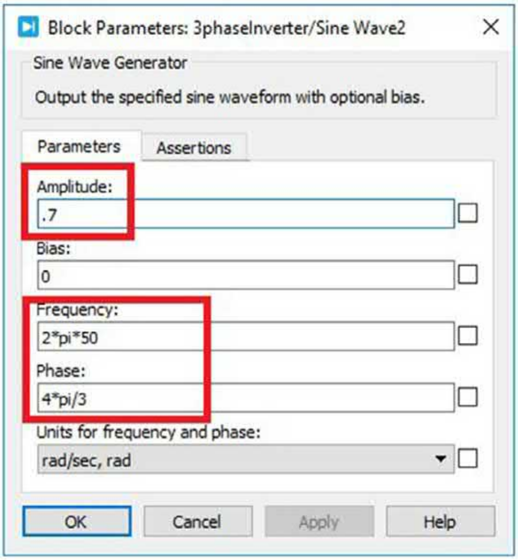
Frequency: 2*pi*50 ☐

Phase: 2*pi/3 ☐

Units for frequency and phase: rad/sec, rad ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.256 "Sine Wave1" block settings (see Fig. 3.247).



Block Parameters: 3phaseInverter/Sine Wave2

Sine Wave Generator

Output the specified sine waveform with optional bias.

Parameters | Assertions

Amplitude: .7 ☐

Bias: 0 ☐

Frequency: 2*pi*50 ☐

Phase: 4*pi/3 ☐

Units for frequency and phase: rad/sec, rad ☐

OK
Cancel
Apply
Help

FIG. 3.257 "Sine Wave2" block settings (see Fig. 3.247).

3.9.1.2 Power Stage

Power stage is shown in Fig. 3.258. It composed of 6 IGBTs. Instead of using 6 separate IGBT blocks, you can use three ready-to-use IGBT legs (see Fig. 3.259).

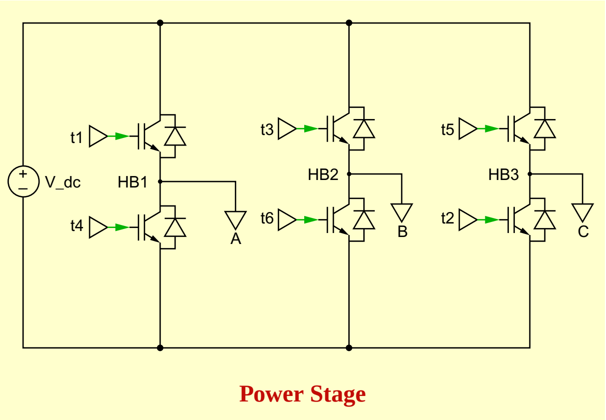
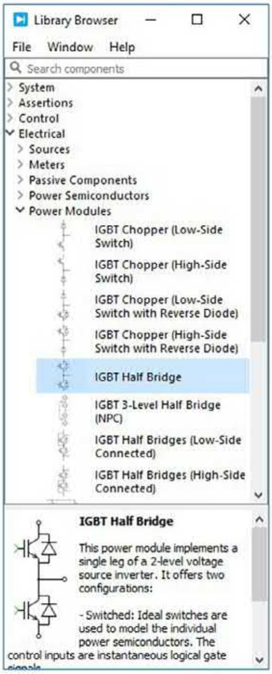

t1
t3
t5
V_dc
HB1
HB2
HB3
t4
t6
t2
A
B
C
Power Stage

FIG. 3.258 Power Stage section of 3-phase inverter.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Power Modules
IGBT Chopper (Low-Side Switch)
IGBT Chopper (High-Side Switch)
IGBT Chopper (Low-Side Switch with Reverse Diode)
IGBT Chopper (High-Side Switch with Reverse Diode)
IGBT Half Bridge
IGBT 3-Level Half Bridge (NPC)
IGBT Half Bridges (Low-Side Connected)
IGBT Half Bridges (High-Side Connected)

IGBT Half Bridge

This power module implements a single leg of a 2-level voltage source inverter. It offers two configurations:

- Switched: Ideal switches are used to model the individual power semiconductors. The control inputs are instantaneous logical gate

FIG. 3.259 IGBT Half-Bridge block.

3.9.1.3 *Load*

Load is shown in Fig. 3.260. It is Y-connected balanced load ($R = 4\,\Omega$ and $L = 5\,\text{mH}$).

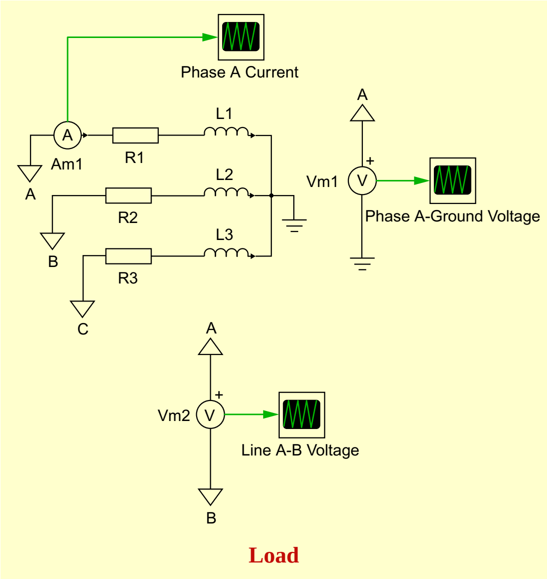

Phase A Current
L1
A
Am1
R1
A
L2
Vm1
V
+
Phase A-Ground Voltage
A
R2
B
L3
R3
C
A
+
Vm2
V
Line A-B Voltage
B
Load

FIG. 3.260 Load section of 3 phase rectifier ($R = 4\,\Omega$ and $L = 5\,\text{mH}$).

Simulation results are shown in Figs. 3.261–3.263.

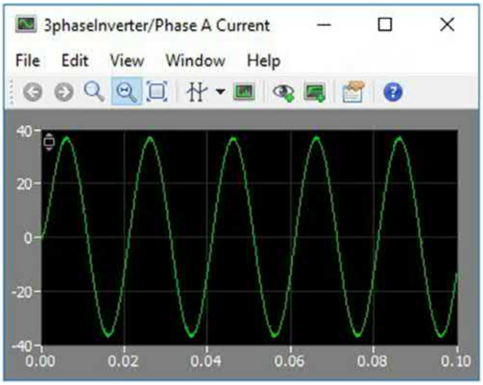

3phaseInverter/Phase A Current
File Edit View Window Help
40
20
0
-20
-40
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 3.261 Phase A current.

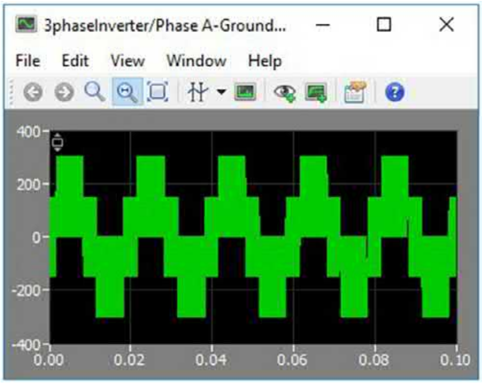

3phaseInverter/Phase A-Ground...
File Edit View Window Help
400
200
0
-200
-400
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 3.262 Phase A-ground voltage.

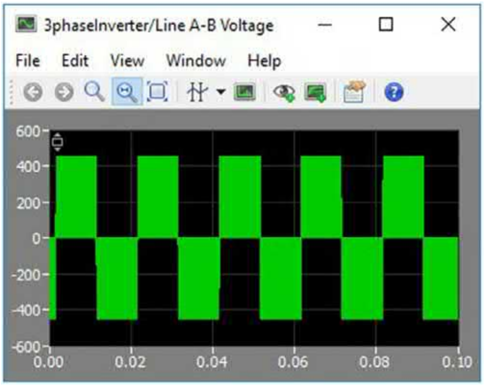
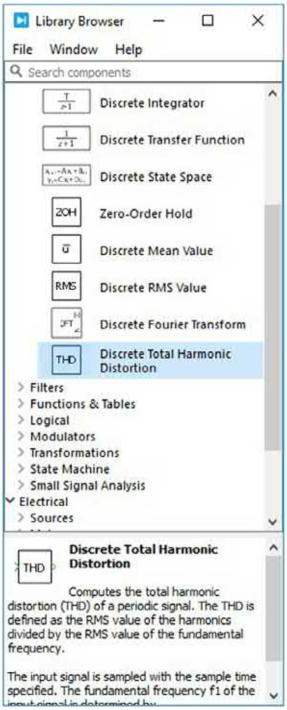

3phaseInverter/Line A-B Voltage
File Edit View Window Help
600
400
200
0
-200
-400
-600
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 3.263 Line-to-line voltage.

3.9.2 Calculating the total harmonic distortion (THD)

PLECS can calculate the THD of a waveform. You can use the "Discrete Total Harmonic Distortion" block to calculate the THD (Fig. 3.264).


Library Browser
File
Window
Help
Search components
Discrete Integrator
Discrete Transfer Function
Discrete State Space
ZOH
Zero-Order Hold
Discrete Mean Value
RMS
Discrete RMS Value
Discrete Fourier Transform
THD
Discrete Total Harmonic Distortion
Filters
Functions & Tables
Logical
Modulators
Transformations
State Machine
Small Signal Analysis
Electrical
Sources
THD

Discrete Total Harmonic Distortion

Computes the total harmonic distortion (THD) of a periodic signal. The THD is defined as the RMS value of the harmonics divided by the RMS value of the fundamental frequency.

The input signal is sampled with the sample time specified. The fundamental frequency f1 of the

FIG. 3.264 "Discrete Total Harmonic Distortion" block.

Before calculating the THD of the 3-phase inverter, we will study a simple example to learn how "Discrete Total Harmonic Distortion" block works. Assume a simulation like that shown in Fig. 3.265.

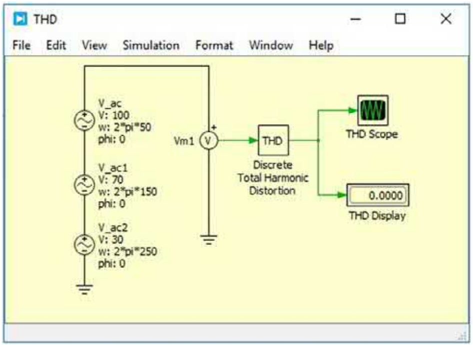
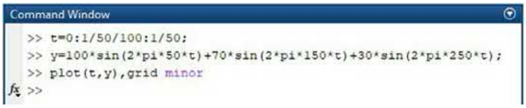

THD
File
Edit
View
Simulation
Format
Window
Help
V_ac
V: 100
w: 2*pi*50
phi: 0
V_ac1
V: 70
w: 2*pi*150
phi: 0
V_ac2
V: 30
w: 2*pi*250
phi: 0
Vm1
V
+
THD
Discrete
Total Harmonic
Distortion
THD Scope
0.0000
THD Display

FIG. 3.265 A simple simulation diagram to understand the working principle of "Discrete Total Harmonic Distortion" block.

According to Fig. 3.265, the input voltage is

$$100\sin\left(2\times\pi\times50\times t\right)+70\sin\left(2\times\pi\times150\times t\right)+30\sin\left(2\times\pi\times250\times t\right)$$

This voltage has fundamental frequency of 50 Hz and contains 3rd and 5th harmonics. This voltage can be drawn easily using MATLAB as shown in Fig. 3.266. Result is shown in Fig. 3.267. As you can see, the waveform does not seems like a pure sinusoidal wave since it contains harmonics. So, we expect a high THD for it.



```
>> t=0:1/50/100:1/50;
>> y=100*sin(2*pi*50*t)+70*sin(2*pi*150*t)+30*sin(2*pi*250*t);
>> plot(t,y),grid minor
fx >>
```

FIG. 3.266 MATLAB code to draw one period of the $100\sin(2\times\pi\times50\times t)+70\sin(2\times\pi\times150\times t)+30\sin(2\times\pi\times250\times t)$.

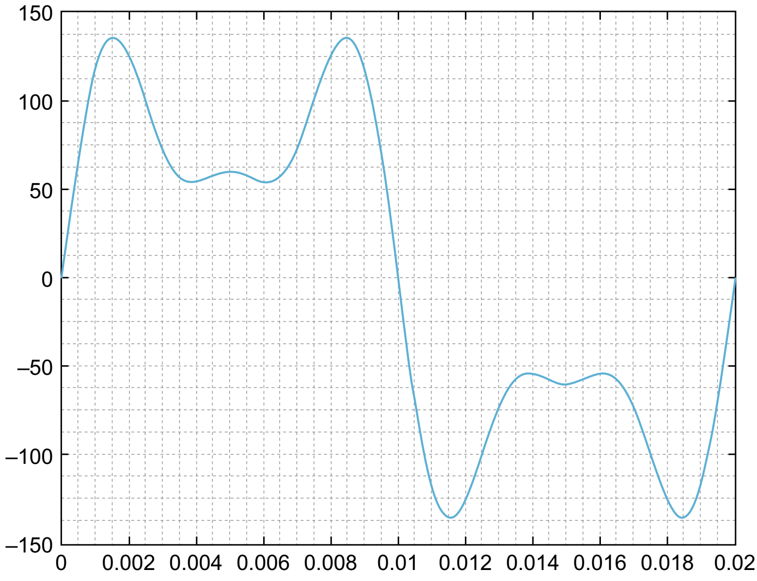


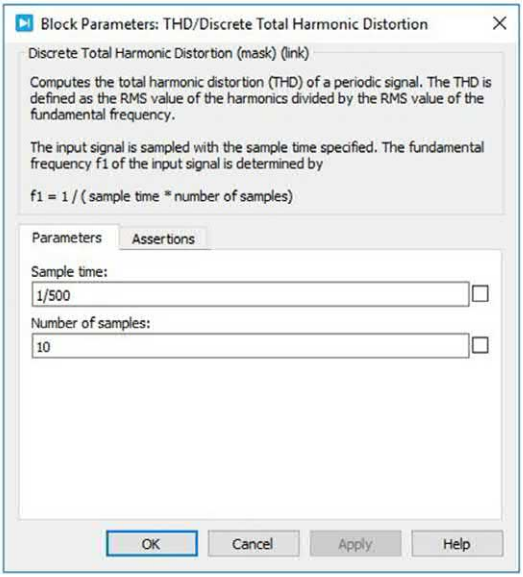
FIG. 3.267 Diagram of $100\sin(2\times\pi\times50\times t)+70\sin(2\times\pi\times150\times t)+30\sin(2\times\pi\times250\times t)$.

THD of input voltage can be calculated easily using the following formula

$$\mathrm{THD}=\frac{\sqrt{\sum_{n=2}^{\infty}{V_n}^2}}{V_1}=\frac{\sqrt{\left(\frac{70}{\sqrt{2}}\right)^2+\left(\frac{30}{\sqrt{2}}\right)^2}}{\frac{100}{\sqrt{2}}}=\frac{53.85}{70.71}=0.76$$

V_n shows RMS value of the nth harmonic.

We want to compare the simulation result with the pencil-and-paper analysis. Double click on the "Discrete Total Harmonic Distortion" block and do the setting as shown in Fig. 3.268.



Discrete Total Harmonic Distortion (mask) (link)

Computes the total harmonic distortion (THD) of a periodic signal. The THD is defined as the RMS value of the harmonics divided by the RMS value of the fundamental frequency.

The input signal is sampled with the sample time specified. The fundamental frequency f1 of the input signal is determined by

f1 = 1 / (sample time * number of samples)

Sample time: 1/500 ☐

Number of samples: 10 ☐

FIG. 3.268 "Discrete Total Harmonic Distortion" block settings.

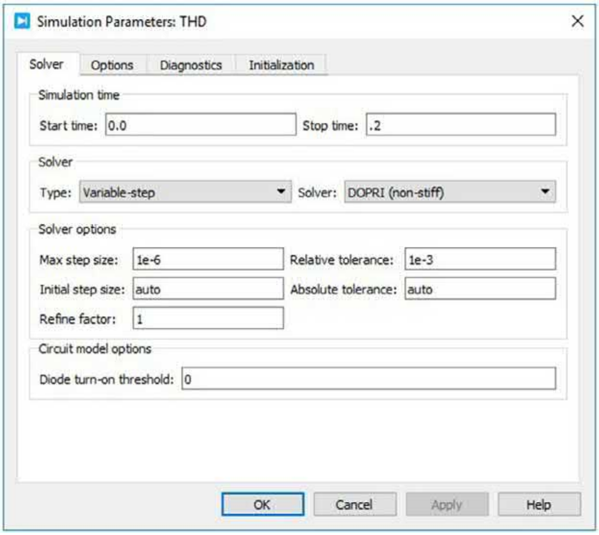
As shown in Fig. 3.268, the fundamental frequency of "Discrete Total Harmonic Distortion" block is calculated with the aid of:

$$f = \frac{1}{\text{Sample time} \times \text{Number of samples}}$$

Since we enter the "Sample time:" box as $\frac{1}{500}$ and "Number of samples:" as 10, the fundamental frequency is calculated as:

$$f = \frac{1}{\frac{1}{500} \times 10} = 50\,\text{Hz}$$

So with these settings, we specify the fundamental frequency correctly. If we run the simulation with the settings shown in Fig. 3.269, we obtain the result shown in Fig. 3.270. As you can see, PLECS calculates the THD as 0.7000, which is close to pencil-and-paper analysis.


Simulation Parameters: THD
Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

Simulation time

Start time: 0.0 Stop time: .2

Solver

Type: Variable-step Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-6 Relative tolerance: 1e-3

Initial step size: auto Absolute tolerance: auto

Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

FIG. 3.269 Simulation Parameters.

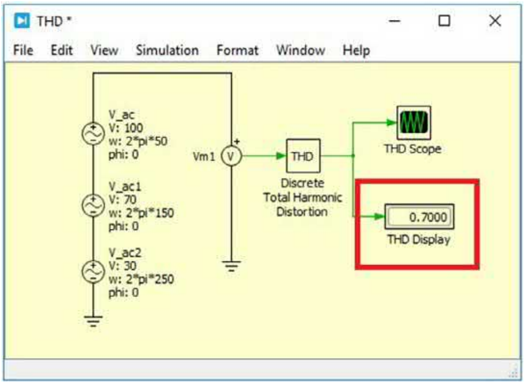

THD *
File
Edit
View
Simulation
Format
Window
Help
V_ac
V: 100
w: 2*pi*50
phi: 0
V_ac1
V: 70
w: 2*pi*150
phi: 0
V_ac2
V: 30
w: 2*pi*250
phi: 0
Vm1
V
+
THD
Discrete
Total Harmonic
Distortion
THD Scope
0.7000
THD Display

FIG. 3.270 With settings shown in Fig. 3.268, THD is calculated as 0.7000.

You can open the "THD Scope" block and see the calculated THD as well. When you use scopes to monitor the output of "Discrete Total Harmonic Distortion" block only, notice the steady-state part of graph (Figs. 3.271 and 3.272). Transients part of graph has no importance.

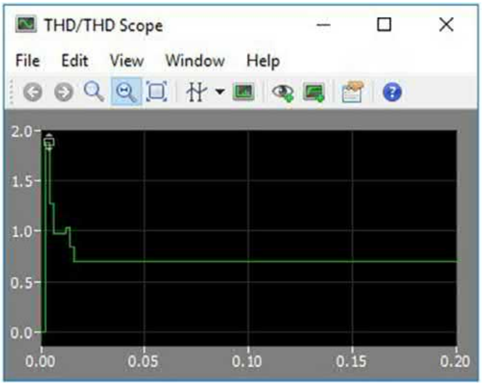
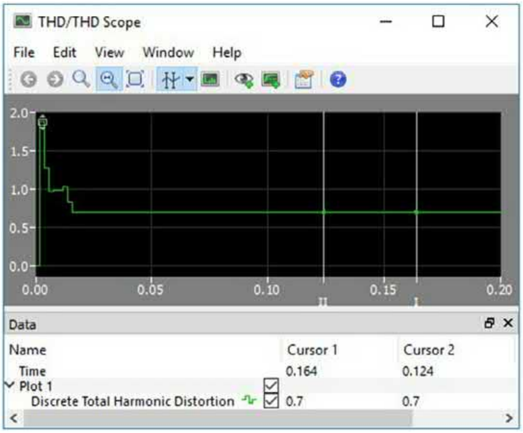

THD/THD Scope
File
Edit
View
Window
Help
2.0
1.5
1.0
0.5
0.0
0.00
0.05
0.10
0.15
0.20

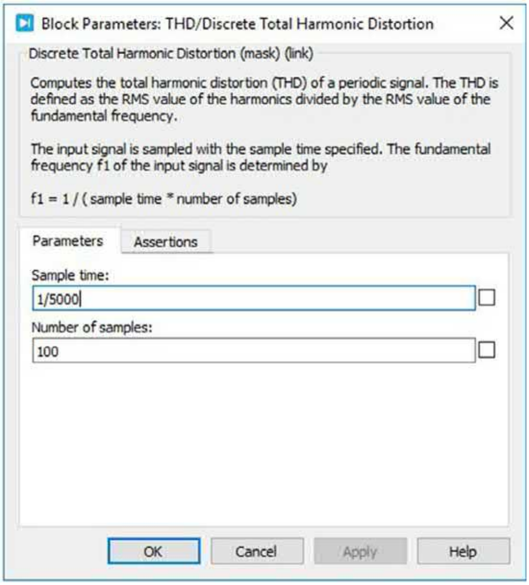
FIG. 3.271 Output of "Discrete Total Harmonic Distortion" block.


THD/THD Scope
File
Edit
View
Window
Help
2.0
1.5
1.0
0.5
0.0
0.00
0.05
0.10
0.15
0.20
II
I
Data

Name		Cursor 1	Cursor 2
Time		0.164	0.124
Plot 1	☑		
Discrete Total Harmonic Distortion	☑	0.7	0.7

FIG. 3.272 Using cursors to read the THD. Place cursor in the steady-state portion of the graph, transients part is not important. THD is calculated as 0.7.

"Number of samples" can be selected as 10 for most of simulations but you can increase it if you need more accuracy. We increase the "Number of samples" to 100 and decrease the "Sample time" to $\frac{1}{5000}$. So, the fundamental frequency is 50 Hz. If we run the simulation with settings shown in Fig. 3.273, the result shown in Fig. 3.274 will be obtained, which is close to the pencil-and-paper analysis.


Block Parameters: THD/Discrete Total Harmonic Distortion
Parameters
Assertions
OK
Cancel
Apply
Help

Discrete Total Harmonic Distortion (mask) (link)

Computes the total harmonic distortion (THD) of a periodic signal. The THD is defined as the RMS value of the harmonics divided by the RMS value of the fundamental frequency.

The input signal is sampled with the sample time specified. The fundamental frequency f1 of the input signal is determined by

f1 = 1 / (sample time * number of samples)

Sample time: 1/5000 ☐

Number of samples: 100 ☐

FIG. 3.273 Increasing the "Number of samples" box to 100.

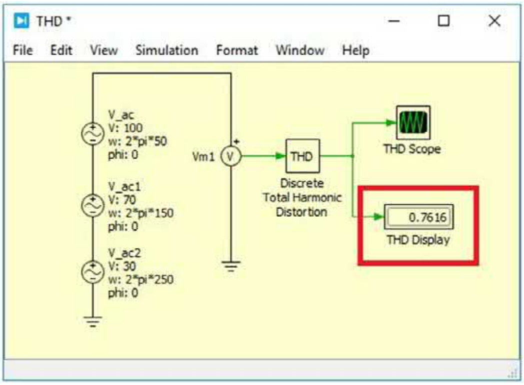

THD *
File
Edit
View
Simulation
Format
Window
Help
V_ac
V: 100
w: 2*pi*50
phi: 0
V_ac1
V: 70
w: 2*pi*150
phi: 0
V_ac2
V: 30
w: 2*pi*250
phi: 0
Vm1
V
+
THD
Discrete
Total Harmonic
Distortion
THD Scope
0.7616
THD Display

FIG. 3.274 With settings shown in Fig. 3.273 THD is calculated as 0.7616.

We are ready to do the THD analysis of 3-phase inverter at this point. We change the schematic to that shown in Fig. 3.275. The "Discrete Total Harmonic Distortion" block settings are the same as Fig. 3.273.

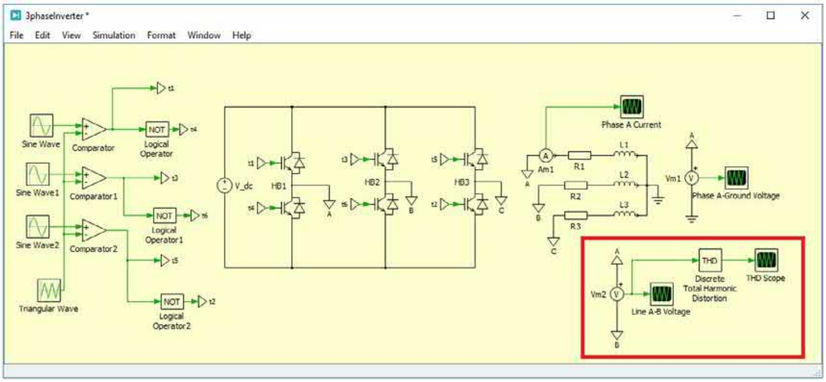

3phaseInverter *
File Edit View Simulation Format Window Help
Sine Wave
Sine Wave1
Sine Wave2
Triangular Wave
Comparator
Comparator1
Comparator2
NOT
Logical Operator
Logical Operator1
Logical Operator2
t1
t2
t3
t4
t5
t6
V_dc
HB1
HB2
HB3
A
B
C
Phase A Current
Am1
R1
R2
R3
L1
L2
L3
Vm1
Phase A-Ground Voltage
Vm2
Line A-B Voltage
THD
Discrete Total Harmonic Distortion
THD Scope

FIG. 3.275 Connecting a "Discrete Total Harmonic Distortion" block to line-line voltage to measure its distortion.

If we run the simulation, we obtain the results shown in Fig. 3.276.

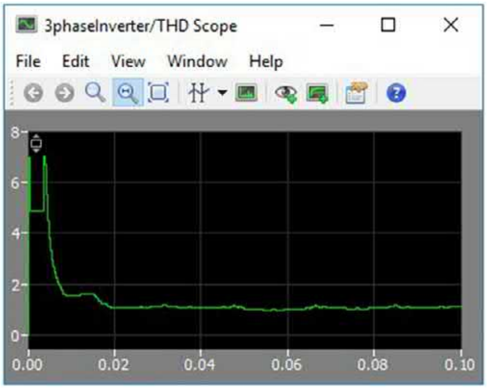


FIG. 3.276 Output of "Discrete Total Harmonic Distortion" block.

You can zoom in the graph to see more details. The THD is about 1.10 (Fig. 3.277).

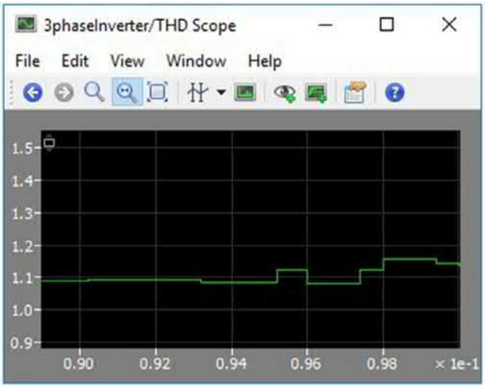


FIG. 3.277 THD is about 1.1.

3.9.3 "Fourier series" block

Simulation diagram shown in Fig. 3.265 and redrawn in Fig. 3.278 for ease of reference can be drawn as shown in Fig. 3.279 using the "Fourier Series" block. Fourier series block can be found in the Functions and Tables section of Library Browser (Fig. 3.280).

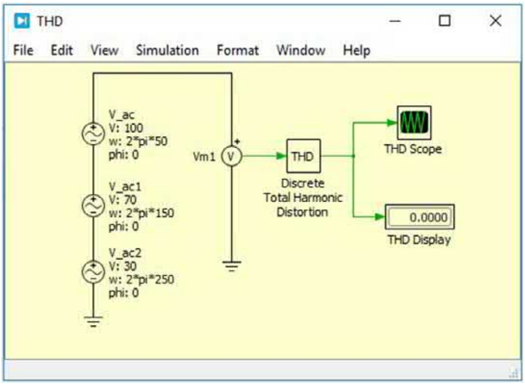

THD
File
Edit
View
Simulation
Format
Window
Help
V_ac
V: 100
w: 2*pi*50
phi: 0
V_ac1
V: 70
w: 2*pi*150
phi: 0
V_ac2
V: 30
w: 2*pi*250
phi: 0
Vm1
V
THD
Discrete
Total Harmonic
Distortion
THD Scope
0.0000
THD Display

FIG. 3.278 Producing the $100\sin(2\times\pi\times50\times t)+70\sin(2\times\pi\times150\times t)+30\sin(2\times\pi\times250\times t)$ using the AC voltage sources.

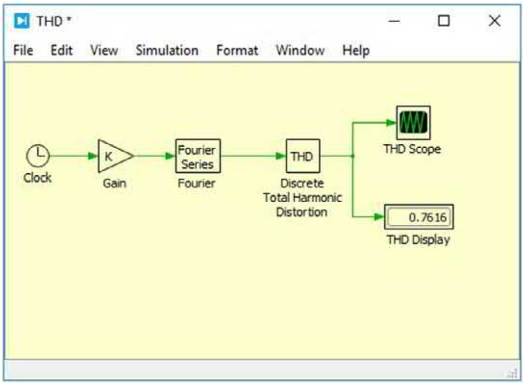
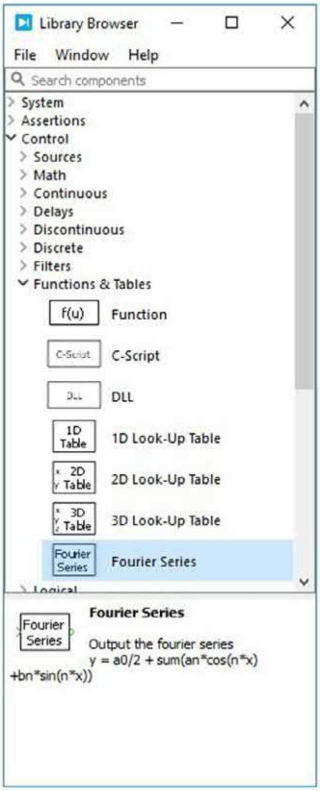

THD *
File
Edit
View
Simulation
Format
Window
Help
Clock
K
Gain
Fourier
Series
Fourier
THD
Discrete
Total Harmonic
Distortion
THD Scope
0.7616
THD Display

FIG. 3.279 Using the Fourier Series block to produce the $100\sin(2\times\pi\times50\times t)+70\sin(2\times\pi\times150\times t)+30\sin(2\times\pi\times250\times t)]$


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
f(u)
Function
C-Script
C-Script
DLL
DLL
1D Table
1D Look-Up Table
2D Table
2D Look-Up Table
3D Table
3D Look-Up Table
Fourier Series
Fourier Series
Logical
Fourier Series

Fourier Series

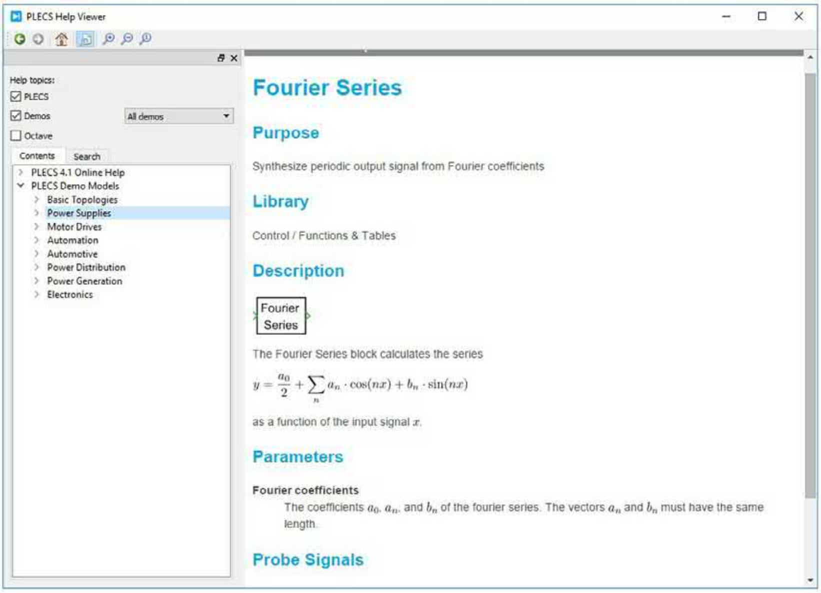
Output the fourier series
y = a0/2 + sum(an*cos(n*x)
+bn*sin(n*x))

FIG. 3.280 Fourier Series block.

Fourier series block takes the coefficients a_n and b_n and produces the output (y) given by:

$$y = \frac{a_0}{2} + \sum_{n=1,2,3,\ldots} a_n \cos(n.x) + \sum_{n=1,2,3,\ldots} b_n \sin(n.x),$$

where x shows the input signal to the block. You can right click on the Fourier block and click the Help to see its description (Fig. 3.281).



PLECS Help Viewer

Help topics:

☑ PLECS

☑ Demos All demos

☐ Octave

- PLECS 4.1 Online Help
- PLECS Demo Models
  - Basic Topologies
  - Power Supplies
  - Motor Drives
  - Automation
  - Automotive
  - Power Distribution
  - Power Generation
  - Electronics

Fourier Series

Purpose

Synthesize periodic output signal from Fourier coefficients

Library

Control / Functions & Tables

Description

The Fourier Series block calculates the series

$$y = \frac{a_0}{2} + \sum_n a_n \cdot \cos(nx) + b_n \cdot \sin(nx)$$

as a function of the input signal x.

Parameters

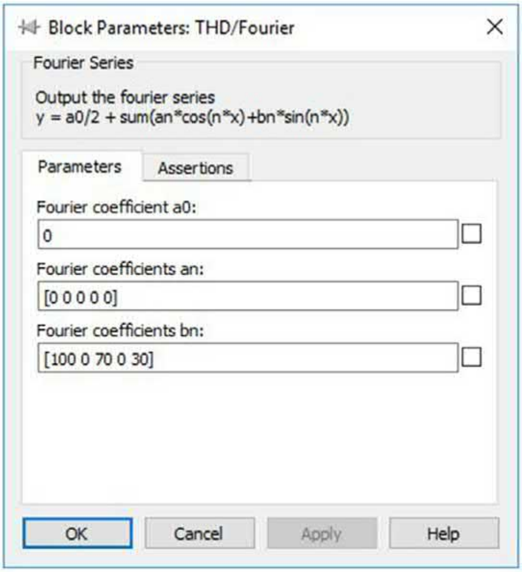
Fourier coefficients

The coefficients a_0, a_n, and b_n of the fourier series. The vectors a_n and b_n must have the same length.

Probe Signals

FIG. 3.281 Fourier Series block documentation.

Double click on the Fourier block and set the parameters as shown in Fig. 3.282.



Block Parameters: THD/Fourier

Fourier Series

Output the fourier series
y = a0/2 + sum(an*cos(n*x)+bn*sin(n*x))

Fourier coefficient a0: 0 ☐

Fourier coefficients an: [0 0 0 0 0] ☐

Fourier coefficients bn: [100 0 70 0 30] ☐

FIG. 3.282 Fourier Series block settings.

Using the values shown in Fig. 3.282, we obtain the following output:

$$y = 100\sin(x) + 70\sin(3x) + 30\sin(5x)$$

If the input to the Fourier series block (x) equal to $2\pi \times 50 \times t$, we can produce the signal $y = 100\sin(2 \times \pi \times 50) + 70\sin(2 \times \pi \times 150) + 30\sin(2 \times \pi \times 250)$, which is exactly the required waveform. One can obtain $x = 2\pi \times 50 \times t$ by passing the clock signal from a constant gain block as shown in Fig. 3.283 (Fig. 3.284).

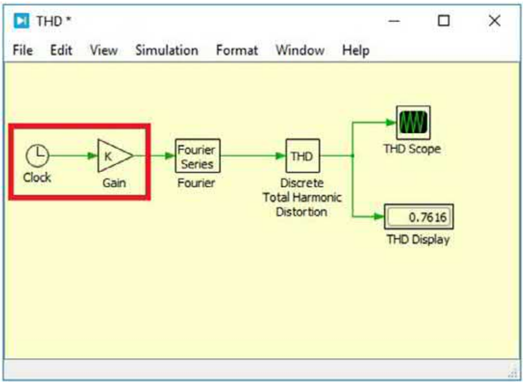
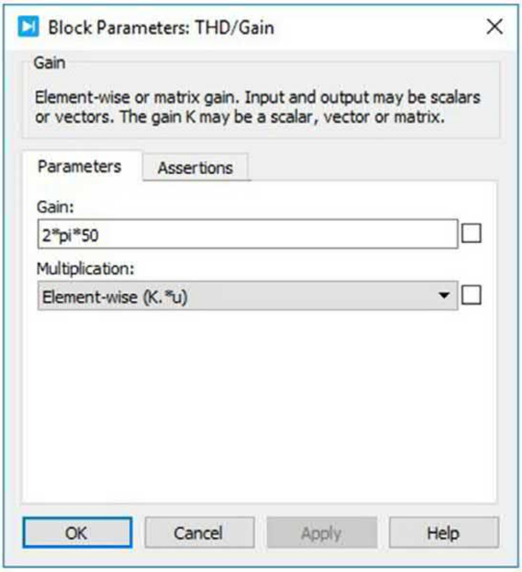

THD *
File
Edit
View
Simulation
Format
Window
Help
Clock
K
Gain
Fourier Series
Fourier
THD
Discrete Total Harmonic Distortion
THD Scope
0.7616
THD Display

FIG. 3.283 Producing the $2 \times \pi \times 50 \times t$ as the input of Fourier Series block. t shows the time. Gain block equals to $2 \times \pi \times 50$.


Block Parameters: THD/Gain
Gain
Parameters
Assertions
OK
Cancel
Apply
Help

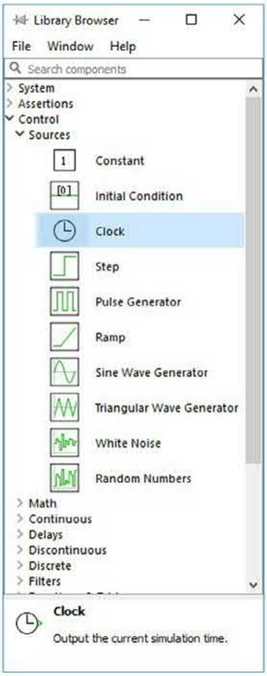
Element-wise or matrix gain. Input and output may be scalars or vectors. The gain K may be a scalar, vector or matrix.

Gain: 2*pi*50 ☐

Multiplication: Element-wise (K.*u) ☐

FIG. 3.284 Gain block settings.

Clock block can be found in the “Sources” section of Library Browser (Fig. 3.285).


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Constant
Initial Condition
Clock
Step
Pulse Generator
Ramp
Sine Wave Generator
Triangular Wave Generator
White Noise
Random Numbers
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Clock

Output the current simulation time.

FIG. 3.285 Clock block.

You can convert the output of Fourier series block into a voltage (which can drive circuits) using a controlled voltage source block. Controlled voltage source block can be found in the Sources section of Library Browser (Figs. 3.286 and 3.287).

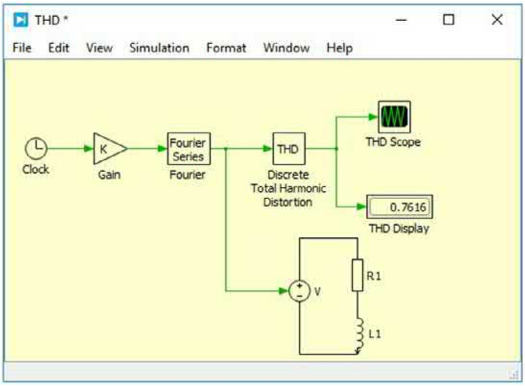
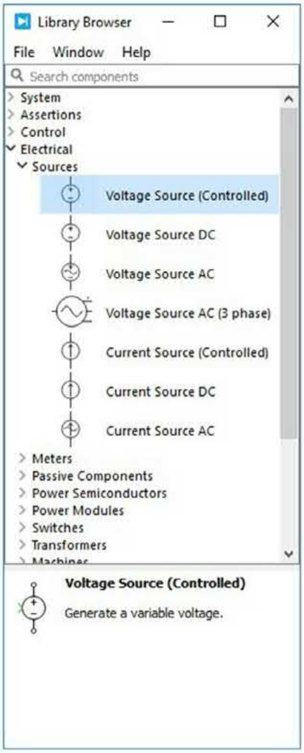

THD *
File
Edit
View
Simulation
Format
Window
Help
Clock
K
Gain
Fourier Series
Fourier
THD
Discrete Total Harmonic Distortion
THD Scope
0.7616
THD Display
V
R1
L1

FIG. 3.286 Conversion of Fourier Series block output into voltage using a controllable voltage source.


Library Browser
File Window Help
Search components
System
Assertions
Control
Electrical
Sources
Voltage Source (Controlled)
Voltage Source DC
Voltage Source AC
Voltage Source AC (3 phase)
Current Source (Controlled)
Current Source DC
Current Source AC
Meters
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines

Voltage Source (Controlled)

Generate a variable voltage.

FIG. 3.287 Voltage Source (Controlled) block.

3.10 Example 3.10: Simulation of electrical machines

PLECS has a powerful library of electrical machines. In this example, we will simulate a DC motor with a variable load.

3.10.1 Preparing the simulation

Simulation of electrical machines is possible in PLECS. PLECS has a power Machines library, which contains models of most electrical machines (Fig. 3.288).

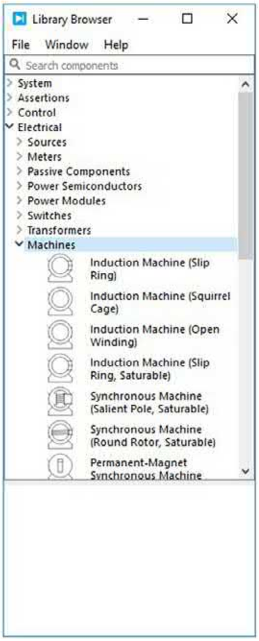

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines
Induction Machine (Slip Ring)
Induction Machine (Squirrel Cage)
Induction Machine (Open Winding)
Induction Machine (Slip Ring, Saturable)
Synchronous Machine (Salient Pole, Saturable)
Synchronous Machine (Round Rotor, Saturable)
Permanent-Magnet Synchronous Machine

FIG. 3.288 Machines section of the Library Browser.

PLECS has mechanical components such as loads, gear boxes, etc. These parts are placed in the "Mechanical" section of Library Browser (Fig. 3.289).

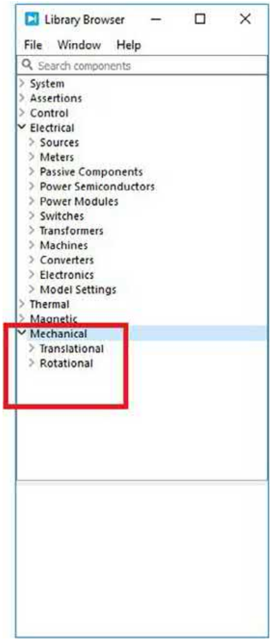

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines
Converters
Electronics
Model Settings
Thermal
Magnetic
Mechanical
Translational
Rotational

FIG. 3.289 Mechanical section of Library Browser. It contains blocks to simulate both translational and rotational systems.

In this section, we study a simple example. Assume a simulation diagram such as one shown in Fig. 3.290. The blocks places used are shown in Figs. 3.291–3.294. The motor torque is 15 N m for $0<t<2$ s and increase to 30 N m or $t>2$ s. We monitor the shaft speed.

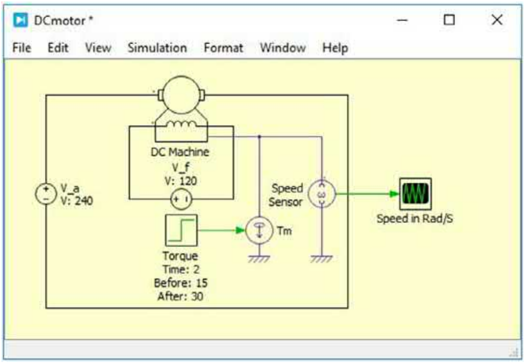
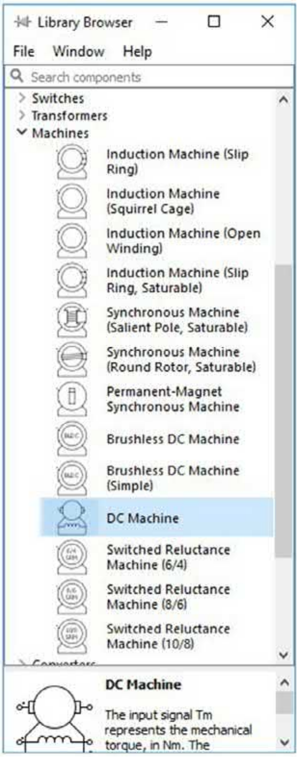
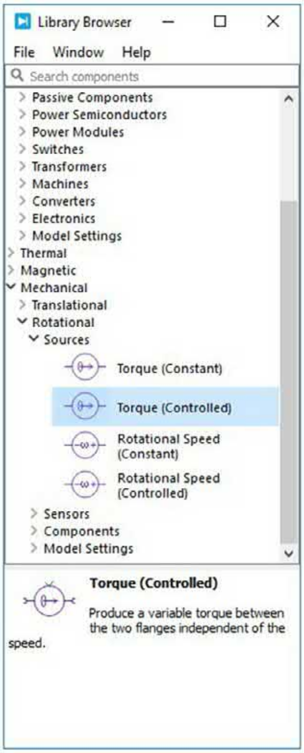

DCmotor *
File
Edit
View
Simulation
Format
Window
Help
DC Machine
V_f
V: 120
V_a
V: 240
Speed
Sensor
Tm
Torque
Time: 2
Before: 15
After: 30
Speed in Rad/S

FIG. 3.290 Simulation of DC machine connected to a variable load.


Library Browser
File
Window
Help
Search components
Switches
Transformers
Machines
Induction Machine (Slip Ring)
Induction Machine (Squirrel Cage)
Induction Machine (Open Winding)
Induction Machine (Slip Ring, Saturable)
Synchronous Machine (Salient Pole, Saturable)
Synchronous Machine (Round Rotor, Saturable)
Permanent-Magnet Synchronous Machine
Brushless DC Machine
Brushless DC Machine (Simple)
DC Machine
Switched Reluctance Machine (6/4)
Switched Reluctance Machine (8/6)
Switched Reluctance Machine (10/8)
Converters
DC Machine

The input signal Tm represents the mechanical torque, in Nm. The

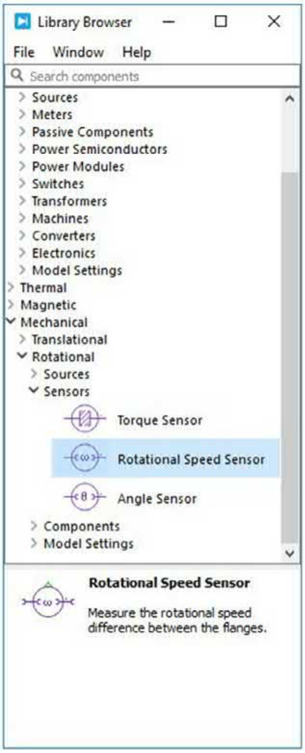
FIG. 3.291 DC machine block.


Library Browser
File
Window
Help
Search components
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines
Converters
Electronics
Model Settings
Thermal
Magnetic
Mechanical
Translational
Rotational
Sources
Torque (Constant)
Torque (Controlled)
Rotational Speed (Constant)
Rotational Speed (Controlled)
Sensors
Components
Model Settings

Torque (Controlled)

Produce a variable torque between the two flanges independent of the speed.

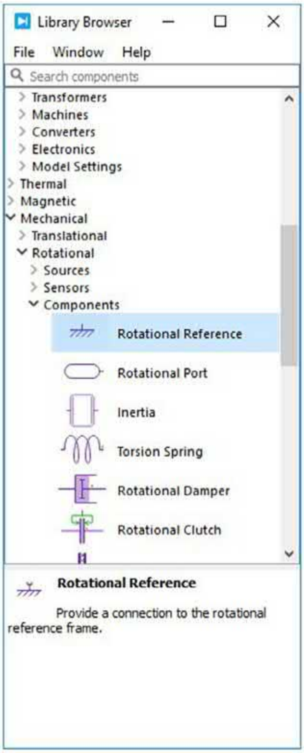
FIG. 3.292 Torque (Controlled) block.


Library Browser
File
Window
Help
Search components
Sources
Meters
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines
Converters
Electronics
Model Settings
Thermal
Magnetic
Mechanical
Translational
Rotational
Sources
Sensors
Torque Sensor
Rotational Speed Sensor
Angle Sensor
Components
Model Settings

Rotational Speed Sensor

Measure the rotational speed difference between the flanges.

FIG. 3.293 Rotational Speed Sensor block.

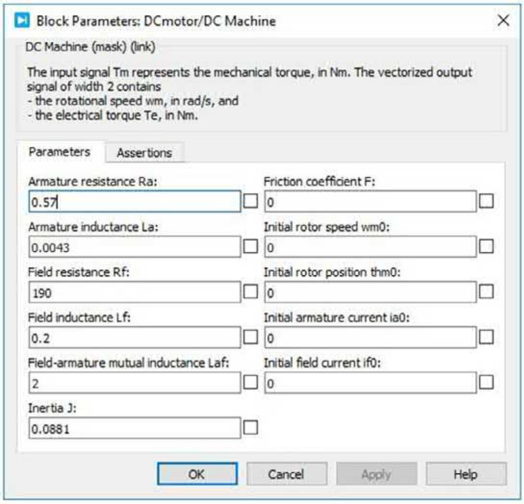

Library Browser
File
Window
Help
Search components
Transformers
Machines
Converters
Electronics
Model Settings
Thermal
Magnetic
Mechanical
Translational
Rotational
Sources
Sensors
Components
Rotational Reference
Rotational Port
Inertia
Torsion Spring
Rotational Damper
Rotational Clutch

Rotational Reference

Provide a connection to the rotational reference frame.

FIG. 3.294 Rotational Reference block.

Blocks settings used are shown in Figs. 3.295 and 3.296.

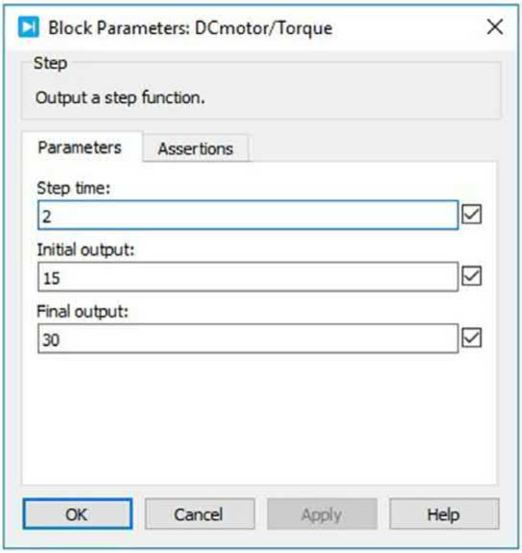

Block Parameters: DCmotor/DC Machine
Parameters
Assertions
OK
Cancel
Apply
Help

DC Machine (mask) (link)

The input signal Tm represents the mechanical torque, in Nm. The vectorized output signal of width 2 contains
- the rotational speed wm, in rad/s, and
- the electrical torque Te, in Nm.

Armature resistance Ra: 0.57 ☐
Armature inductance La: 0.0043 ☐
Field resistance Rf: 190 ☐
Field inductance Lf: 0.2 ☐
Field-armature mutual inductance Laf: 2 ☐
Inertia J: 0.0881 ☐
Friction coefficient F: 0 ☐
Initial rotor speed wm0: 0 ☐
Initial rotor position thm0: 0 ☐
Initial armature current ia0: 0 ☐
Initial field current if0: 0 ☐

FIG. 3.295 DC Machine settings.


Block Parameters: DCmotor/Torque

Step

Output a step function.

Step time: 2 ☑

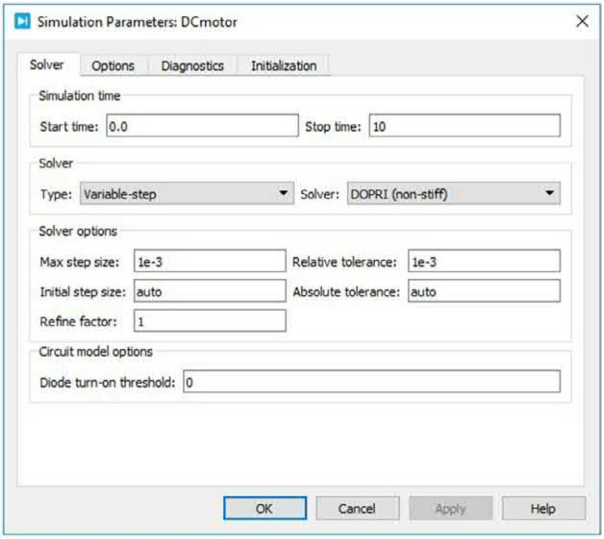
Initial output: 15 ☑

Final output: 30 ☑

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 3.296 Torque block settings.

If we run the simulation with settings shown in Fig. 3.297, we obtain the results shown in Fig. 3.298.


Simulation Parameters: DCmotor

Simulation time

Start time: 0.0

Stop time: 10

Solver

Type: Variable-step

Solver: DOPRI (non-stiff)

Solver options

Max step size: 1e-3

Relative tolerance: 1e-3

Initial step size: auto

Absolute tolerance: auto

Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

Solver
Options
Diagnostics
Initialization
OK
Cancel
Apply
Help

FIG. 3.297 Simulation Parameter.

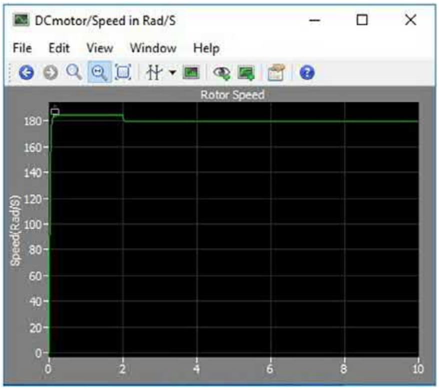
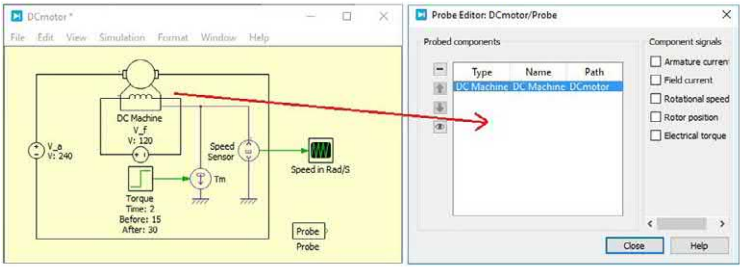


FIG. 3.298 Rotor speed graph.

3.10.2 Monitoring using the probe block

You can use probe block to monitor the desired output as well. For example, assume that we want to see the armature current and shaft speed using the probe block. To do this, place a probe block on the schematic, double click on it to open it, and drag and drop the electric machine inside it (Fig. 3.299).

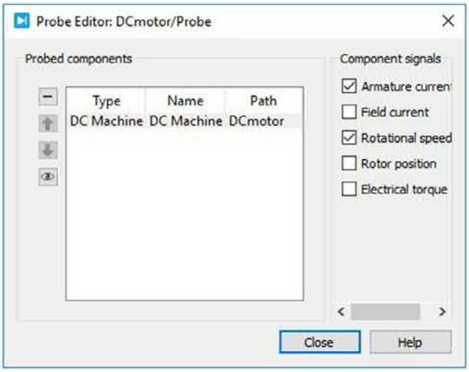

DCmotor *
File
Edit
View
Simulation
Format
Window
Help
DC Machine
V_f
V: 120
V_a
V: 240
Speed
Sensor
Tm
Speed in Rad/S
Torque
Time: 2
Before: 15
After: 30
Probe
Probe
Probe Editor: DCmotor/Probe
Probed components
Component signals
Close
Help

Type	Name	Path
DC Machine	DC Machine	DCmotor

☐ Armature curren
☐ Field current
☐ Rotational speed
☐ Rotor position
☐ Electrical torque

FIG. 3.299 Dragging and dropping the DC Machine into the probe block.

Check the "Armature current" and "Rotational speed" boxes (Fig. 3.300).


Probe Editor: DCmotor/Probe
Close
Help

Probed components

Type	Name	Path
DC Machine	DC Machine	DCmotor

Component signals

☑ Armature curren
☐ Field current
☑ Rotational speed
☐ Rotor position
☐ Electrical torque

FIG. 3.300 Probe block settings.

Connect the output of probe block to a demultiplexer and two scope blocks as shown in Fig. 3.301. Demultiplexer block can be found in the "System" section of Library Browser (Fig. 3.302).

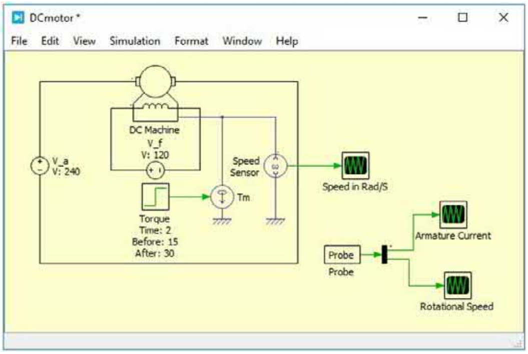
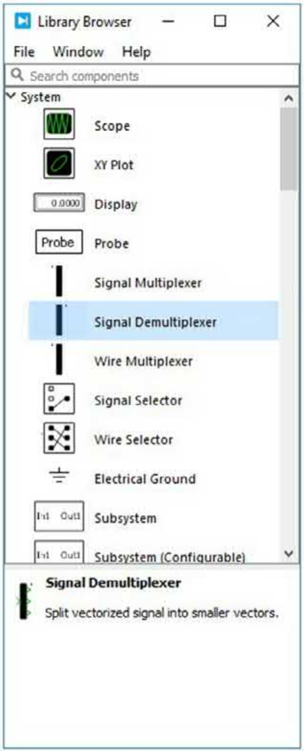

DCmotor *
File Edit View Simulation Format Window Help
DC Machine
V_f
V: 120
V_a
V: 240
Speed
Sensor
Tm
Torque
Time: 2
Before: 15
After: 30
Speed in Rad/S
Armature Current
Probe
Probe
Rotational Speed

FIG. 3.301 Connecting the probe block to a demultiplexer and scope blocks.


Library Browser
File Window Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
Wire Selector
Electrical Ground
In1 Out1
Subsystem
In1 Out1
Subsystem (Configurable)

Signal Demultiplexer

Split vectorized signal into smaller vectors.

FIG. 3.302 Signal Demultiplexer block.

If we run the simulation, we obtain the results shown in Figs. 3.303 and 3.304. As expected, machine speed decreases and the drawn current increases.

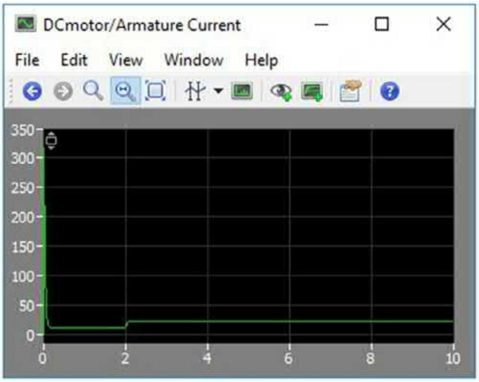


FIG. 3.303 Armature current graph.

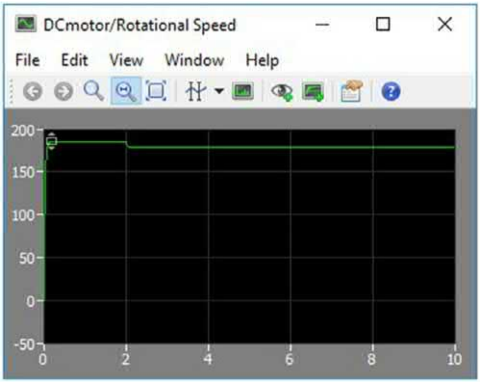


FIG. 3.304 Shaft speed graph.

References

[1] D. Hart, Power Electronics, Mc Graw Hill, 2011.

Further reading

[2] P. Krein, Elements of Power Electronics, Oxford University Press, 2014.
[3] N. Mohan, T. Undeland, Power Electronics Converters, Applications and Design, Wiley, 2002.
[4] M. Rashid, Power Electronics Devices, Circuits and Applications, Pearson, 2013.
[5] R. Erikson, D. Maksimovic, Fundamentals of Power Electronics, Springer, 2001.
[6] S. Ang, A. Oliva, Power Switching Converters, Taylor & Francis, 2005.
[7] M.K. Kazimierczuck, Pulse Width Modulated DC-DC Power Converters, John Wiley, 2012.
[8] K. Ogata, Matlab for Control Engineers, Pearson, 2007.

Chapter 4

Simulink® version of PLECS®

Chapter outline

4.1 Introduction 219
4.2 Simulation of diode-clamped inverter 219
4.3 Simulation of a diode-clamped multilevel inverter 224
  4.3.1 The power stage 226
  4.3.2 The PWM generation part 234
  4.3.3 Simulation of circuit 240
4.4 Sending/receiving signals to/from Simulink environment 241
4.5 Simulation of a cascaded inverter 247
4.6 Measurement with the probe block 261
4.7 Extraction of frequency response of DC-DC converters 265
4.8 Fitting a transfer function to obtained graph 273
4.9 Designing a controller 276
4.10 Obtaining the control-to-inductor current transfer function 284
4.11 Extraction of output impedance 285
4.12 Steady-state analysis 289
4.13 More simulation examples 295
Further reading 297

4.1 Introduction

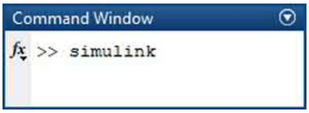
We studied the standalone version of PLECS in the previous chapters. Simulink version of PLECS is the focus of this chapter. The Simulink version uses the Simulink solver to simulate the circuits. Working with Simulink version of PLECS is quite easy and similar to the standalone version.

Using the Simulink version of PLECS, you have access to all the Simulink blocks. So, the Simulink power is added to the PLECS power. For instance, assume that you want to simulate a converter, which has a fuzzy logic controller. You can simply use the Fuzzy Logic Toolbox® blocks and see the results readily. Such a simulation is not an easy task in standalone version. So, ensure to choose right tool for your problem.

4.2 Simulation of diode-clamped inverter

The user must enter the Simulink environment of MATLAB in order to access the PLECS block set. Type `simulink` in the command window to enter the Simulink environment (Fig. 4.1).

Simulation of Power Electronics Converters Using PLECS®. https://doi.org/10.1016/B978-0-12-817364-0.00004-7
© 2020 Elsevier Inc. All rights reserved.


Command Window

```
>> simulink
```

FIG. 4.1 Entering the Simulink environment.

The PLECS block sets are added to the Simulink Library Browser window (Fig. 4.2).

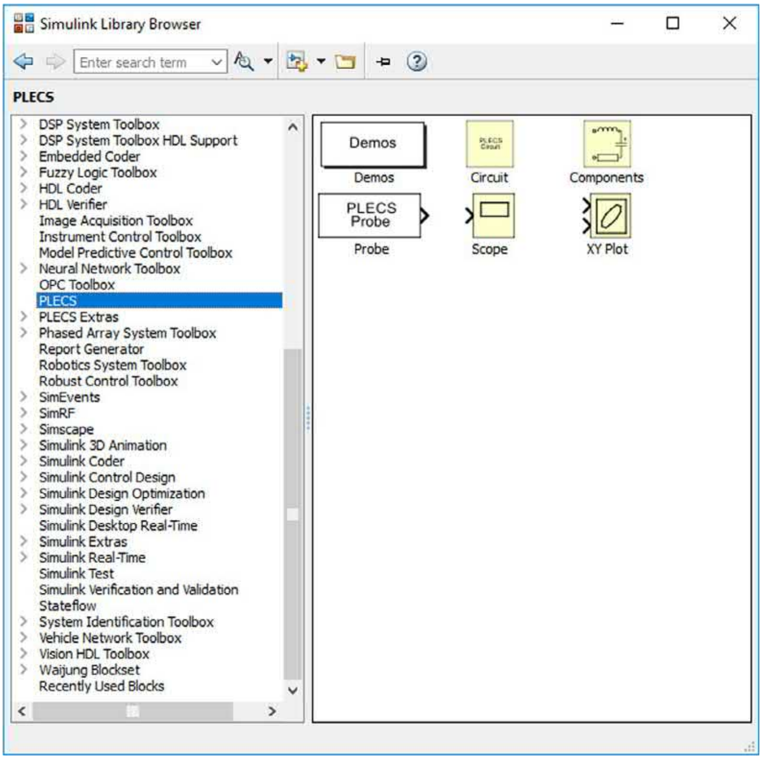

Simulink Library Browser
Enter search term
PLECS
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Desktop Real-Time
Simulink Extras
Simulink Real-Time
Simulink Test
Simulink Verification and Validation
Stateflow
System Identification Toolbox
Vehicle Network Toolbox
Vision HDL Toolbox
Waijung Blockset
Recently Used Blocks
Demos
Demos
Circuit
Components
PLECS Probe
Probe
Scope
XY Plot

FIG. 4.2 The PLECS blockset.

In order to simulate a power electronics converter, you must add a Circuit block to your simulation (Fig. 4.3).

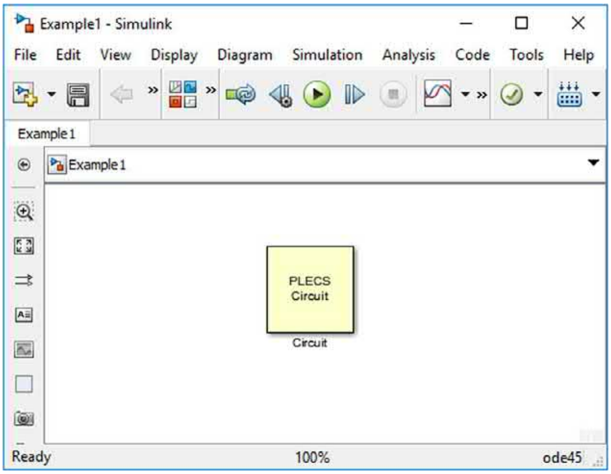

Example1 - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example1
Example1
PLECS
Circuit
Circuit
Ready
100%
ode45

FIG. 4.3 The PLECS circuit block.

Double click the PLECS Circuit block placed in the Simulink environment. The window shown in Fig. 4.4 will appear. As you see, the simulation environment is quite similar to the standalone version.

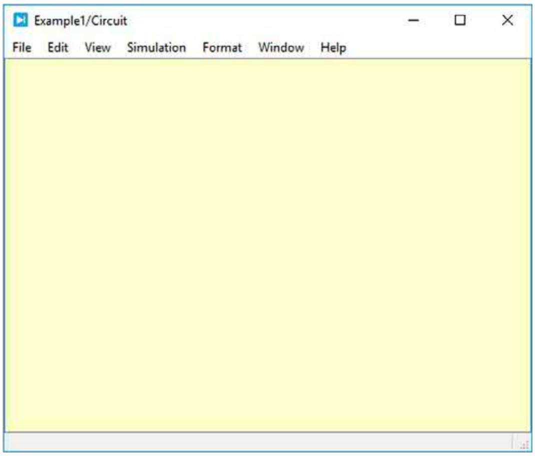

Example1/Circuit
File Edit View Simulation Format Window Help

FIG. 4.4 Inside the PLECS circuit block. You cannot use blocks from other Simulink libraries inside this area. Only blocks provided by PLECS block set can be used inside the PLECS circuit block.

You can import a file produced by standalone version of PLECS using the File menu (Fig. 4.5).

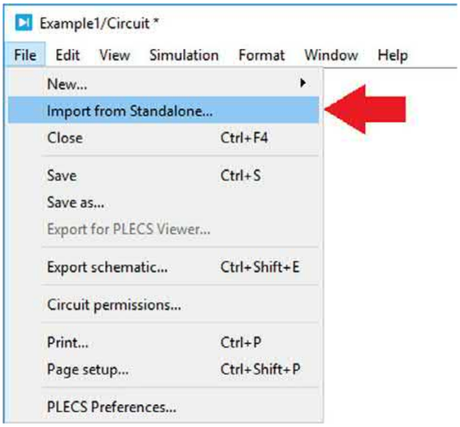

Example1/Circuit *
File
Edit
View
Simulation
Format
Window
Help
New...
Import from Standalone...
Close Ctrl+F4
Save Ctrl+S
Save as...
Export for PLECS Viewer...
Export schematic... Ctrl+Shift+E
Circuit permissions...
Print... Ctrl+P
Page setup... Ctrl+Shift+P
PLECS Preferences...

FIG. 4.5 Importing a file produced by the PLECS Standalone.

Like the standalone version, the components used in the simulation are selected from the Library Browser window (Fig. 4.6).

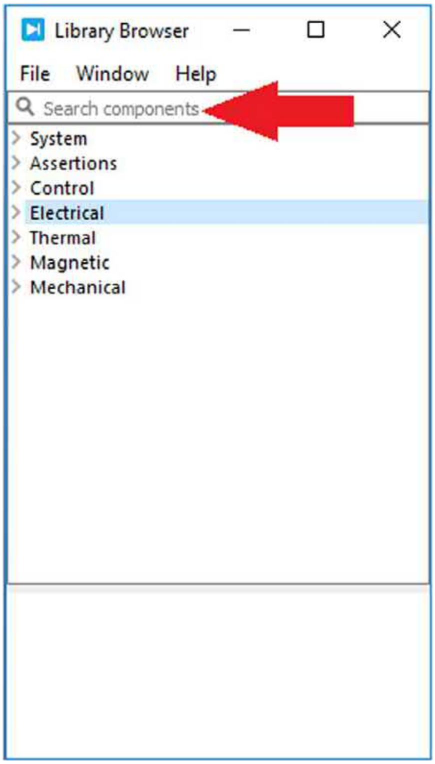

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Thermal
Magnetic
Mechanical

FIG. 4.6 Library Browser window. You can search for a block by typing its name inside the Search components box.

Use the Window menu to appear the Library Browser window (Fig. 4.7).

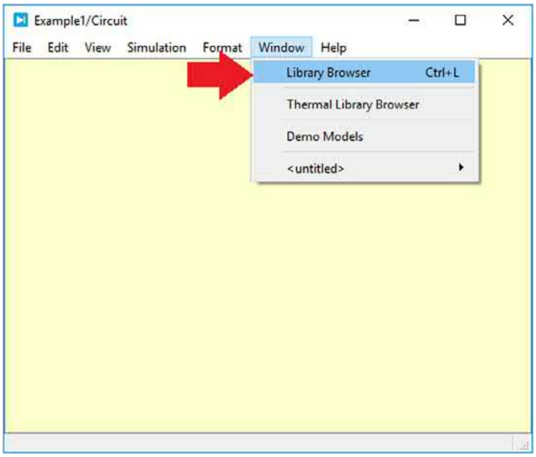

Example1/Circuit
File
Edit
View
Simulation
Format
Window
Help
Library Browser Ctrl+L
Thermal Library Browser
Demo Models
<untitled>

FIG. 4.7 Click the Library Browser window to make the Library Browser window appear.

You can click the Components icon (Fig. 4.8) to appear the Library Browser window, as well.

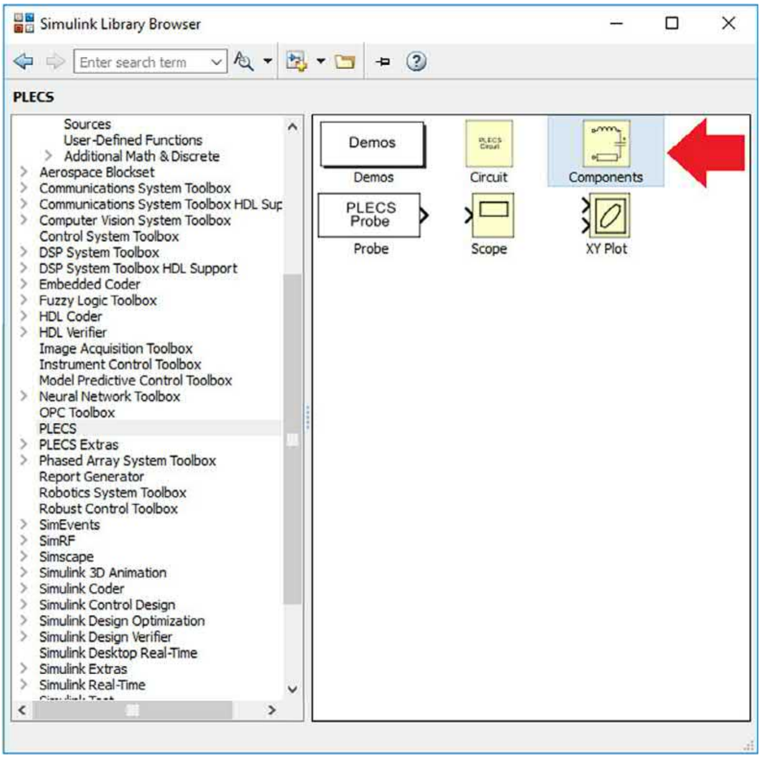

Simulink Library Browser
Enter search term
PLECS
Sources
User-Defined Functions
Additional Math & Discrete
Aerospace Blockset
Communications System Toolbox
Communications System Toolbox HDL Sup
Computer Vision System Toolbox
Control System Toolbox
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Desktop Real-Time
Simulink Extras
Simulink Real-Time
Demos
Demos
Circuit
Components
PLECS Probe
Probe
Scope
XY Plot

FIG. 4.8 The Components section of PLECS block set.

4.3 Simulation of a diode-clamped multilevel inverter

Simulation of a diode-clamped multilevel inverter is studied as the first example. The simulation file is shown in Fig. 4.9.

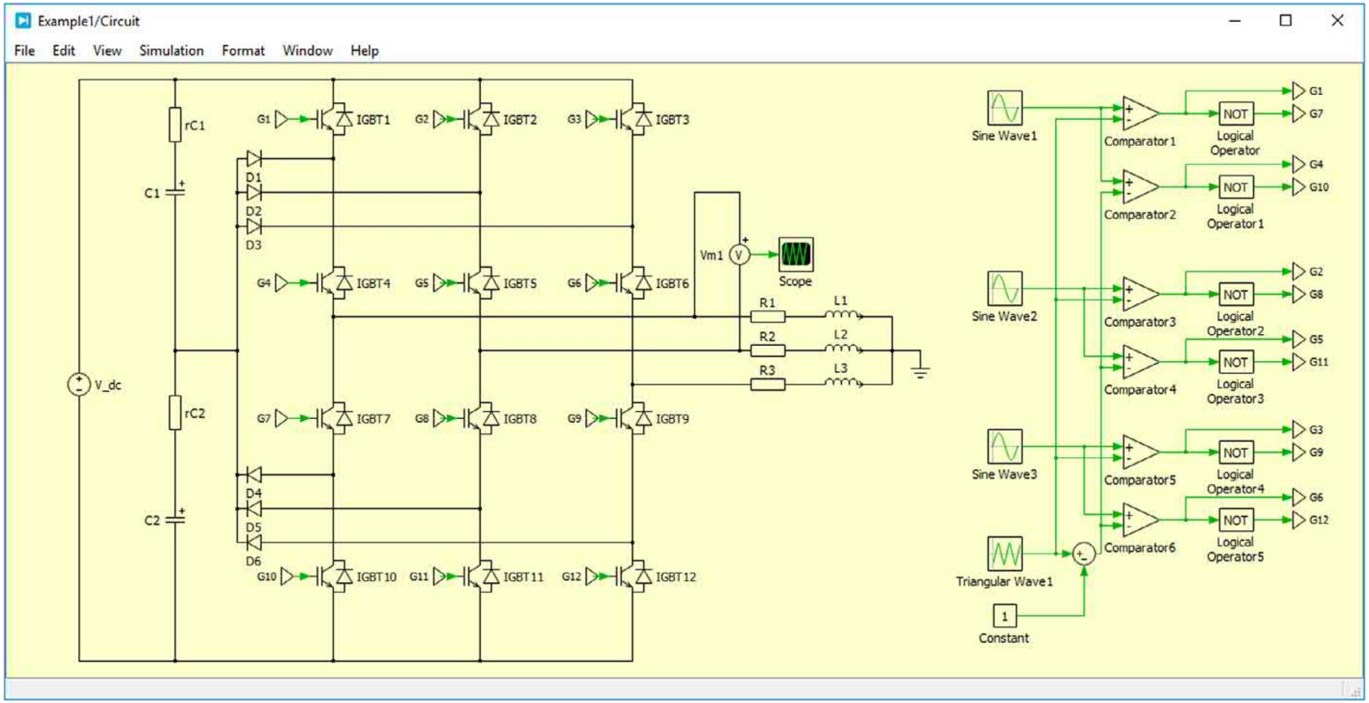

Example1/Circuit
File Edit View Simulation Format Window Help
V_dc
rC1
C1
rC2
C2
D1
D2
D3
D4
D5
D6
G1 IGBT1
G2 IGBT2
G3 IGBT3
G4 IGBT4
G5 IGBT5
G6 IGBT6
G7 IGBT7
G8 IGBT8
G9 IGBT9
G10 IGBT10
G11 IGBT11
G12 IGBT12
Vm1
V
Scope
R1
R2
R3
L1
L2
L3
Sine Wave1
Sine Wave2
Sine Wave3
Triangular Wave1
1
Constant
Comparator1
Comparator2
Comparator3
Comparator4
Comparator5
Comparator6
NOT
Logical Operator
Logical Operator1
Logical Operator2
Logical Operator3
Logical Operator4
Logical Operator5
G1
G7
G4
G10
G2
G8
G5
G11
G3
G9
G6
G12

FIG. 4.9 Schematic of diode-clamped multilevel inverter.

The prepared simulation file is composed of a power stage (Fig. 4.10) and a PWM generation section (Fig. 4.11).

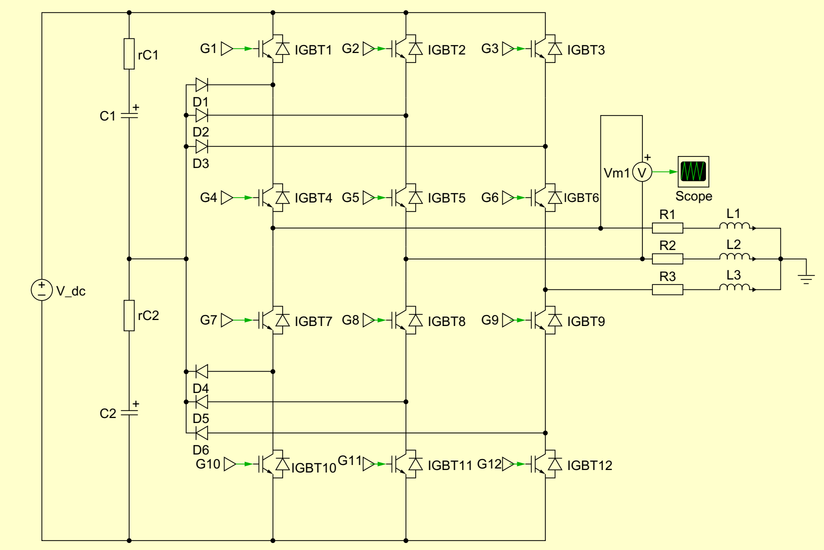

rC1
C1
G1
IGBT1
G2
IGBT2
G3
IGBT3
D1
D2
D3
Vm1
V
Scope
G4
IGBT4
G5
IGBT5
G6
IGBT6
R1
L1
R2
L2
R3
L3
V_dc
rC2
G7
IGBT7
G8
IGBT8
G9
IGBT9
D4
C2
D5
D6
G10
IGBT10
G11
IGBT11
G12
IGBT12

FIG. 4.10 Power stage of diode-clamped multilevel inverter.

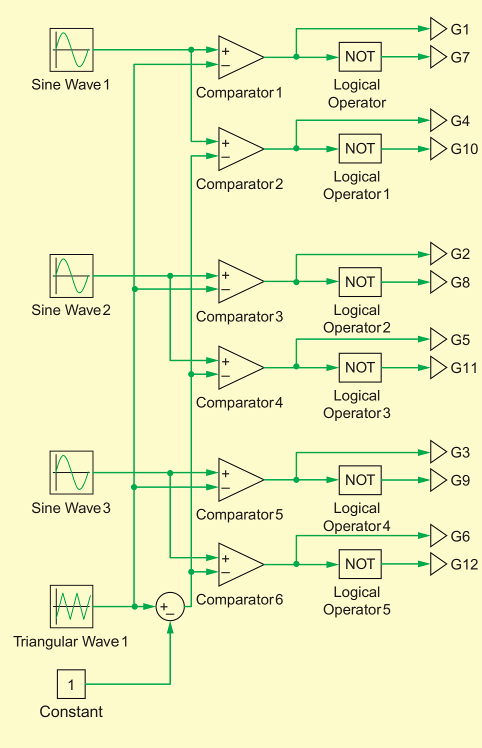

Sine Wave 1
Comparator 1
NOT
Logical Operator
G1
G7
Comparator 2
NOT
Logical Operator 1
G4
G10
Sine Wave 2
Comparator 3
NOT
Logical Operator 2
G2
G8
Comparator 4
NOT
Logical Operator 3
G5
G11
Sine Wave 3
Comparator 5
NOT
Logical Operator 4
G3
G9
Comparator 6
NOT
Logical Operator 5
G6
G12
Triangular Wave 1
1
Constant

FIG. 4.11 PWM generation section of diode-clamped multilevel inverter.

Production of PWM signal for multilevel inverters requires more than one carrier (Fig. 4.12).

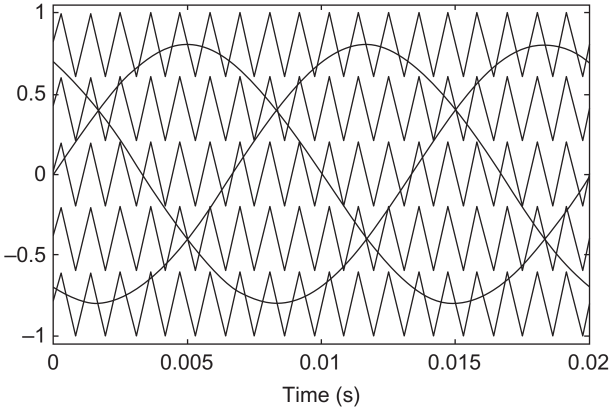


FIG. 4.12 Production of PWM signal for a three-phase diode-clamped multilevel inverter with 11 levels of output. There are five carriers.

The required carriers for the studied diode-clamped inverter (Fig. 4.9) are shown in Fig. 4.13.

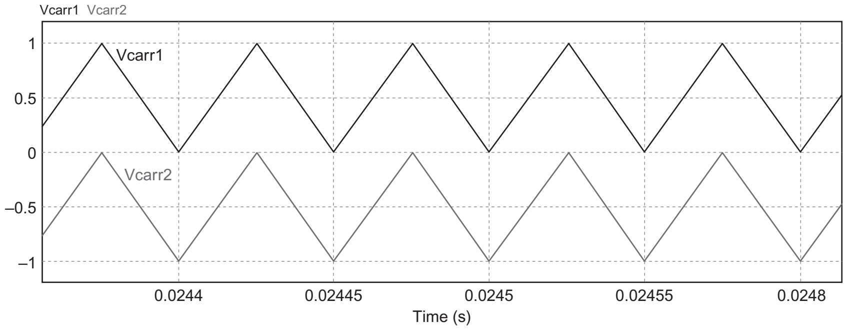
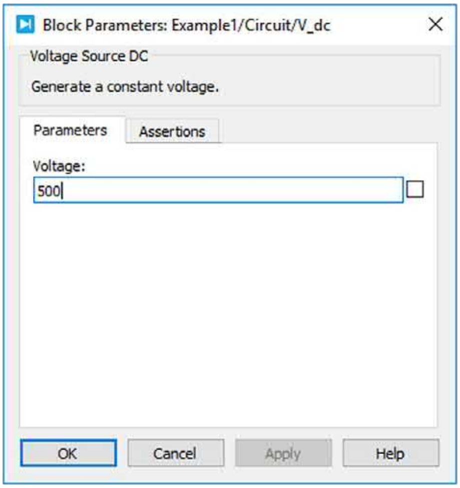


FIG. 4.13 Carriers for production of PWM signal for a diode-clamped multilevel inverter with 5 levels of output. There are two carriers. The reference signal is not shown in this figure.

4.3.1 The power stage

Values of power stage components are shown in Figs. 4.14–4.20.


Block Parameters: Example1/Circuit/V_dc

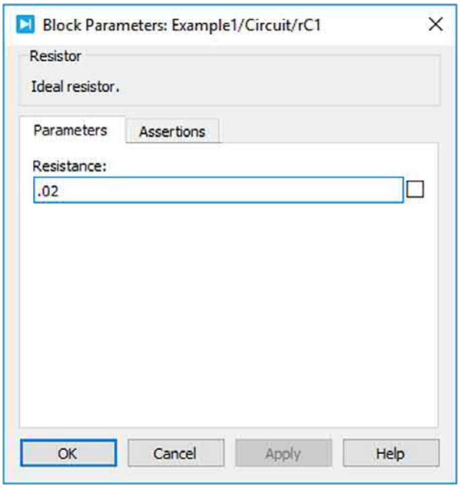
Voltage Source DC

Generate a constant voltage.

Voltage: 500 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.14 Input DC source settings.


Block Parameters: Example1/Circuit/rC1

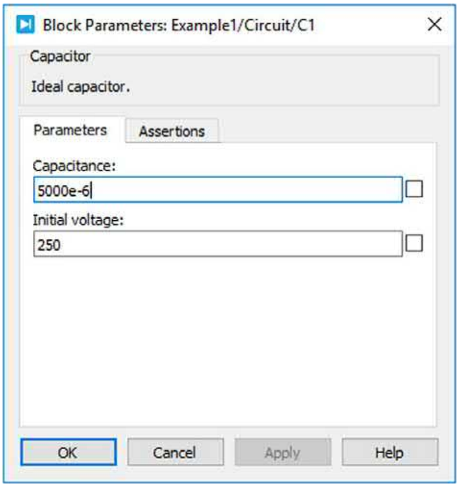
Resistor

Ideal resistor.

Resistance: .02 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.15 Resistor rC1 settings. The resistor rC2 has the same settings.


Block Parameters: Example1/Circuit/C1

Capacitor

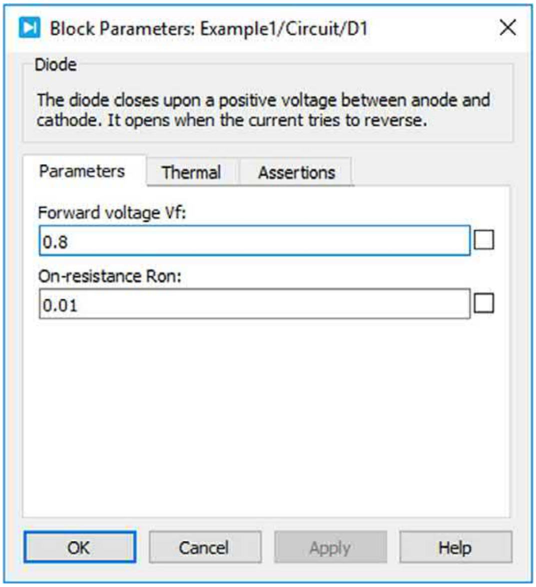
Ideal capacitor.

Capacitance: 5000e-6 ☐

Initial voltage: 250 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.16 Capacitor C1 settings. The capacitor C2 has the same settings.


Block Parameters: Example1/Circuit/D1

Diode

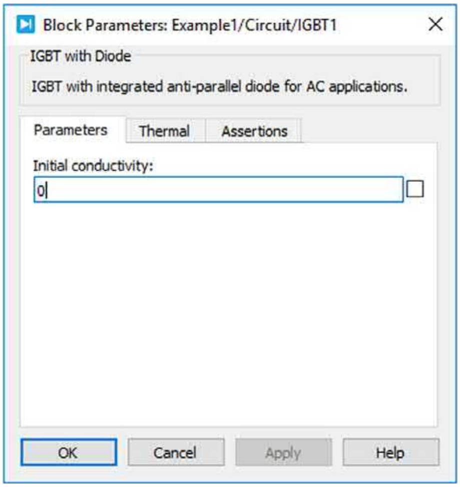
The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

Forward voltage Vf: 0.8 ☐

On-resistance Ron: 0.01 ☐

Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

FIG. 4.17 Diode D1 settings. Other diodes have the same settings.


Block Parameters: Example1/Circuit/IGBT1

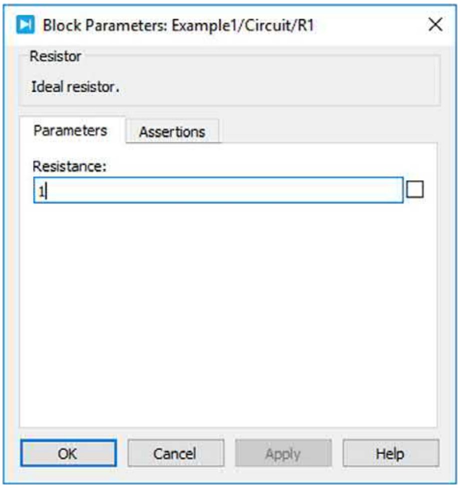
IGBT with Diode

IGBT with integrated anti-parallel diode for AC applications.

Initial conductivity: 0 ☐

Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

FIG. 4.18 IGBT1 settings. Other IGBTs have the same settings.



Block Parameters: Example1/Circuit/R1

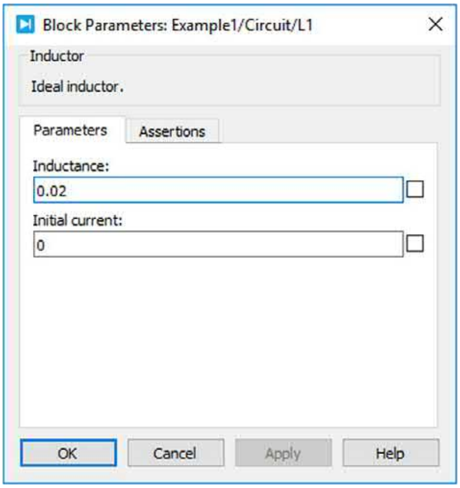
Resistor

Ideal resistor.

Resistance: 1 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.19 Resistor R1 settings. Resistors R2 and R3 have the same settings.



Block Parameters: Example1/Circuit/L1

Inductor

Ideal inductor.

Inductance: 0.02 ☐

Initial current: 0 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.20 Inductor L1 settings. Inductors L2 and L3 have the same settings.

The line-line voltage is measured with the aid of a voltmeter. The voltmeter can be found in the Meters section of Library Browser window (Fig. 4.21).

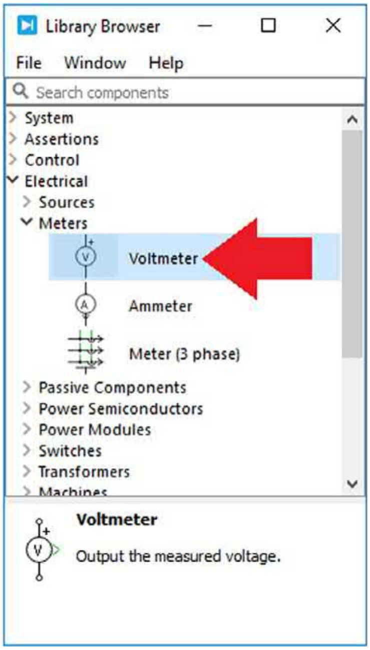

Library Browser
File Window Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Voltmeter
Ammeter
Meter (3 phase)
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines
Voltmeter
Output the measured voltage.

FIG. 4.21 Voltmeter block.

The power stage has 12 Signal From blocks. The Signal From block can be found in the System section of library browser window (Figs. 4.22 and 4.23).

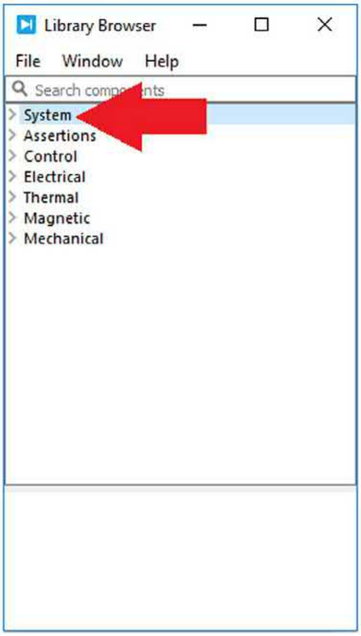
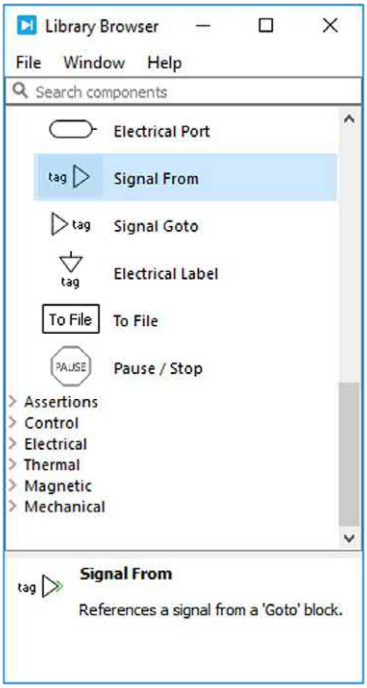

Library Browser
File Window Help
Search components
System
Assertions
Control
Electrical
Thermal
Magnetic
Mechanical

FIG. 4.22 System section of Library Browser window.

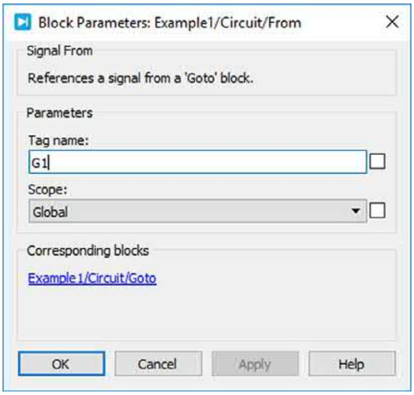

Library Browser
File
Window
Help
Search components
Electrical Port
tag
Signal From
tag
Signal Goto
tag
Electrical Label
To File
To File
PAUSE
Pause / Stop
Assertions
Control
Electrical
Thermal
Magnetic
Mechanical
tag

Signal From

References a signal from a 'Goto' block.

FIG. 4.23 Signal From block.

The Tag name box of Signal From blocks is filled with G1, G2, G3, …, G12 (see Fig. 4.24). The Gi is the Signal From block connected to the gate of i-th IGBT.


Block Parameters: Example1/Circuit/From
Parameters
Corresponding blocks
OK
Cancel
Apply
Help

Signal From

References a signal from a 'Goto' block.

Tag name: G1 ☐

Scope: Global ☐

Example1/Circuit/Goto

FIG. 4.24 The Signal From block G1 settings. The Signals From blocks G2, G3, G4, …, G12 have G2, G3, G4, …, G12 in their Tag name box, respectively.

In fact, the simulation can be done without using the Signal Goto and Signal From blocks. However, the schematic becomes quite messy and un-understandable. Fig. 4.9 is redrawn in Fig. 4.25 for ease of reference. Compare Fig. 4.25 with Fig. 4.26. Fig. 4.26 is more structured and understandable than Fig. 4.25.

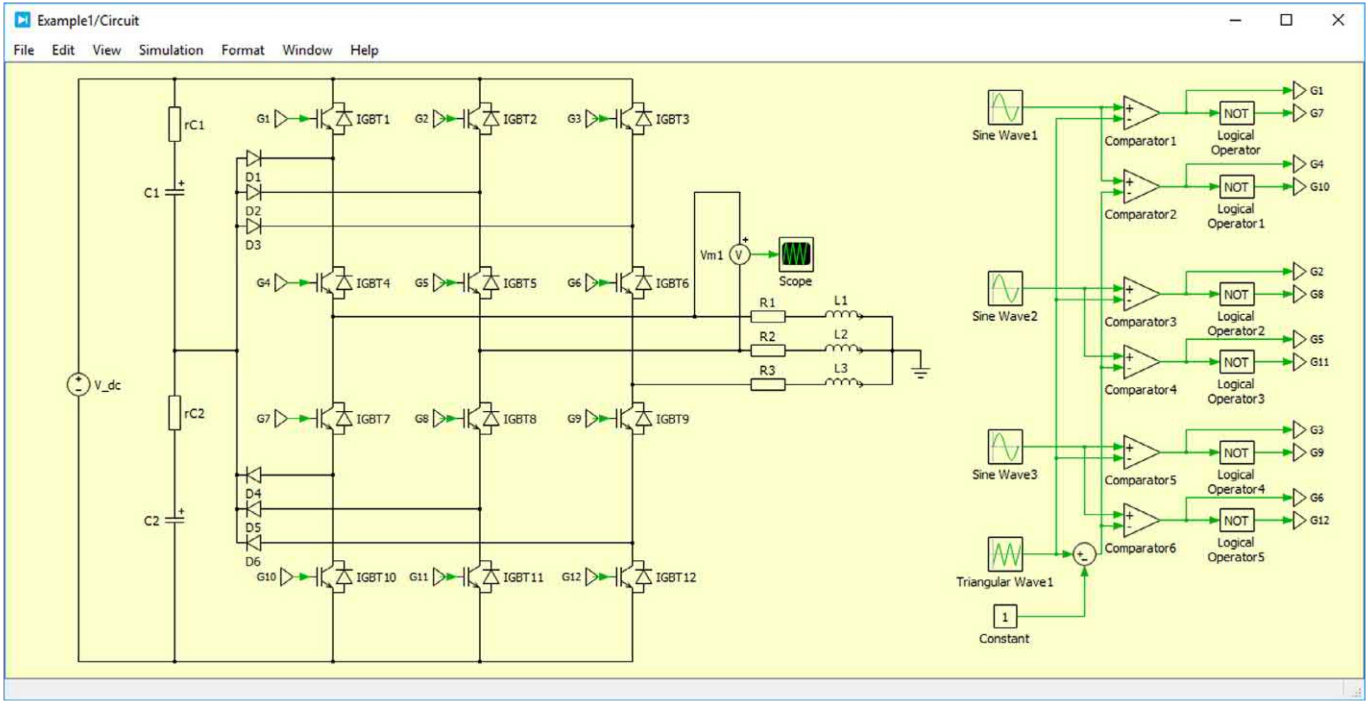

Example1/Circuit
File
Edit
View
Simulation
Format
Window
Help
V_dc
rC1
C1
rC2
C2
D1
D2
D3
D4
D5
D6
G1
IGBT1
G2
IGBT2
G3
IGBT3
G4
IGBT4
G5
IGBT5
G6
IGBT6
G7
IGBT7
G8
IGBT8
G9
IGBT9
G10
IGBT10
G11
IGBT11
G12
IGBT12
Vm1
V
Scope
R1
R2
R3
L1
L2
L3
Sine Wave1
Sine Wave2
Sine Wave3
Triangular Wave1
1
Constant
Comparator1
Comparator2
Comparator3
Comparator4
Comparator5
Comparator6
NOT
Logical Operator
Logical Operator1
Logical Operator2
Logical Operator3
Logical Operator4
Logical Operator5
G1
G7
G4
G10
G2
G8
G5
G11
G3
G9
G6
G12

FIG. 4.25 A good schematic example.

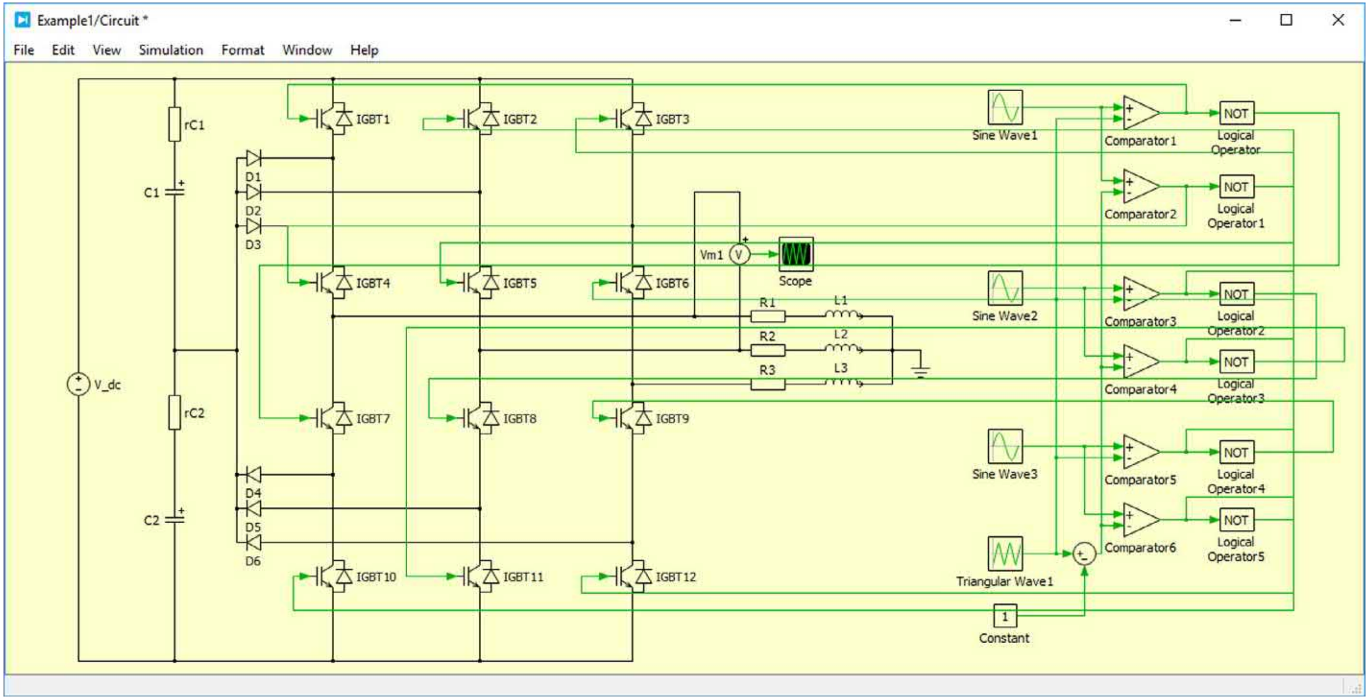

Example1/Circuit *
File
Edit
View
Simulation
Format
Window
Help
rC1
C1
V_dc
rC2
C2
D1
D2
D3
D4
D5
D6
IGBT1
IGBT2
IGBT3
IGBT4
IGBT5
IGBT6
IGBT7
IGBT8
IGBT9
IGBT10
IGBT11
IGBT12
Vm1
V
Scope
R1
R2
R3
L1
L2
L3
Sine Wave1
Sine Wave2
Sine Wave3
Triangular Wave1
Constant
1
Comparator1
Comparator2
Comparator3
Comparator4
Comparator5
Comparator6
NOT
Logical Operator
Logical Operator1
Logical Operator2
Logical Operator3
Logical Operator4
Logical Operator5

FIG. 4.26 A messy and difficult-to-understand schematic. Avoid such schematics.

4.3.2 The PWM generation part

The PWM generation section of the simulation file is reshown in Fig. 4.27.

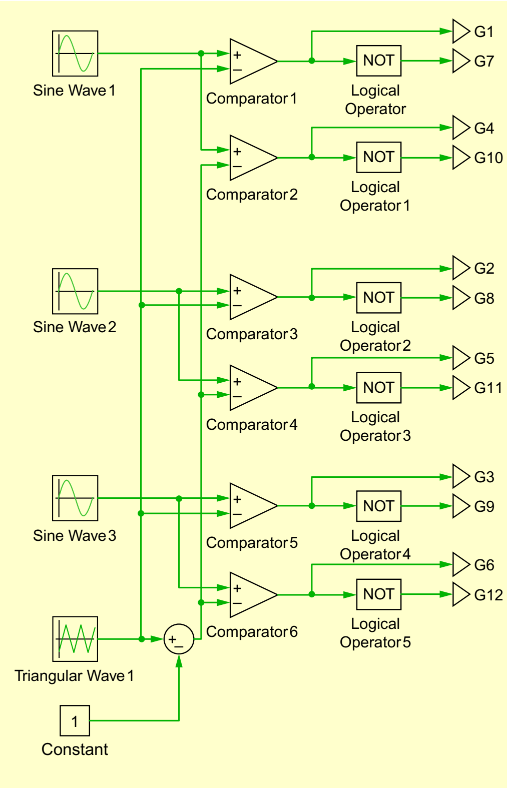
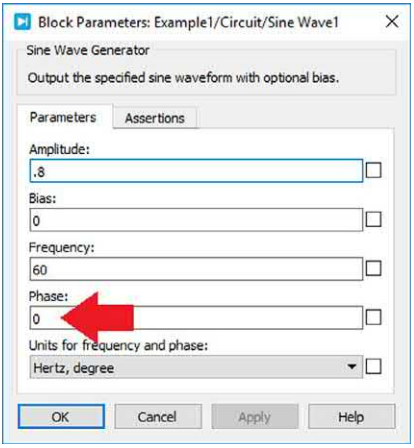

Sine Wave1
Comparator1
NOT
Logical Operator
G1
G7
Comparator2
NOT
Logical Operator1
G4
G10
Sine Wave2
Comparator3
NOT
Logical Operator2
G2
G8
Comparator4
NOT
Logical Operator3
G5
G11
Sine Wave3
Comparator5
NOT
Logical Operator4
G3
G9
Comparator6
NOT
Logical Operator5
G6
G12
Triangular Wave1
1
Constant

FIG. 4.27 The PWM generation section.

Settings of the used blocks are shown in Figs. 4.28–4.32.


Block Parameters: Example1/Circuit/Sine Wave1

Sine Wave Generator

Output the specified sine waveform with optional bias.

Parameters
Assertions

Amplitude: .8

Bias: 0

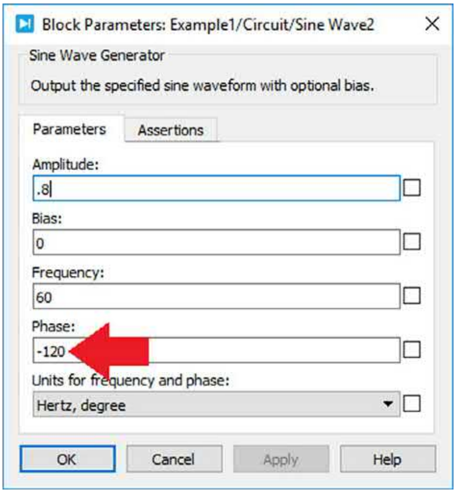
Frequency: 60

Phase: 0

Units for frequency and phase: Hertz, degree

OK
Cancel
Apply
Help

FIG. 4.28 The Sine Wave1 block settings.



Block Parameters: Example1/Circuit/Sine Wave2

Sine Wave Generator

Output the specified sine waveform with optional bias.

Amplitude: .8

Bias: 0

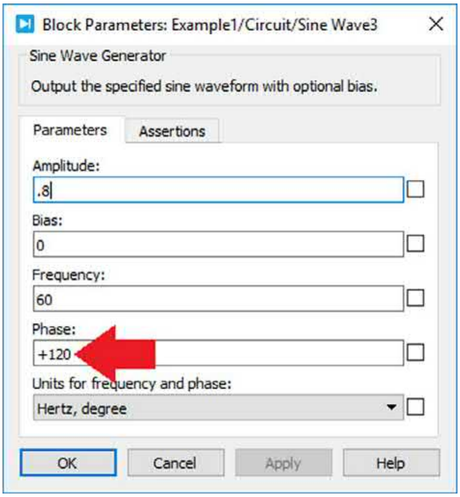
Frequency: 60

Phase: -120

Units for frequency and phase: Hertz, degree

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.29 The Sine Wave2 block settings.



Block Parameters: Example1/Circuit/Sine Wave3

Sine Wave Generator

Output the specified sine waveform with optional bias.

Amplitude: .8

Bias: 0

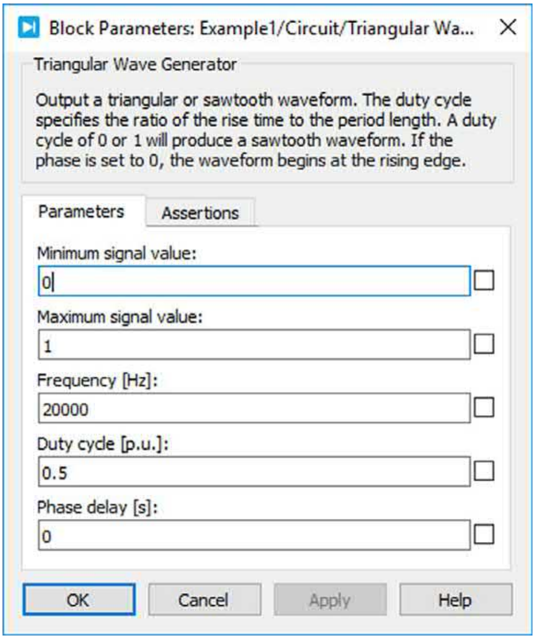
Frequency: 60

Phase: +120

Units for frequency and phase: Hertz, degree

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.30 The Sine Wave3 block settings.



Block Parameters: Example1/Circuit/Triangular Wa...

Triangular Wave Generator

Output a triangular or sawtooth waveform. The duty cycle specifies the ratio of the rise time to the period length. A duty cycle of 0 or 1 will produce a sawtooth waveform. If the phase is set to 0, the waveform begins at the rising edge.

Minimum signal value: 0 ☐

Maximum signal value: 1 ☐

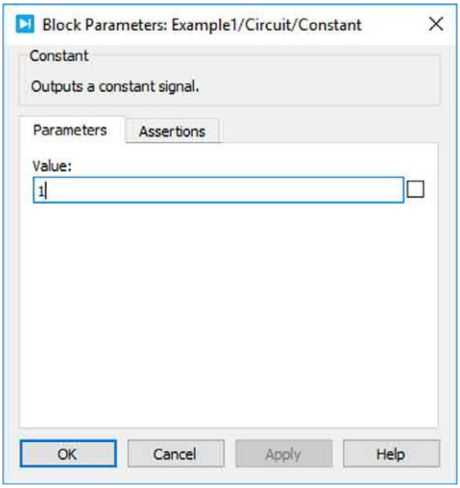
Frequency [Hz]: 20000 ☐

Duty cycle [p.u.]: 0.5 ☐

Phase delay [s]: 0 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.31 The Triangular Wave block settings.



Block Parameters: Example1/Circuit/Constant

Constant

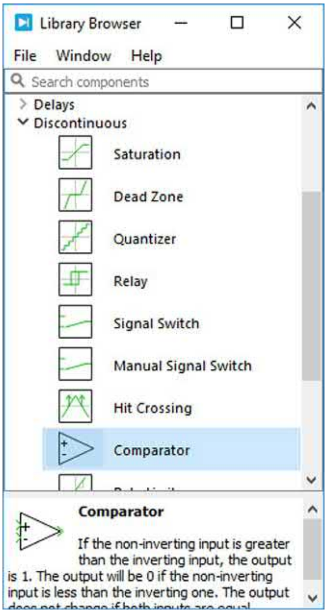
Outputs a constant signal.

Value: 1 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.32 The Constant block settings.

The PWM generation section uses 6 comparators and not blocks. The comparator block can be found in the Discontinuous section of Library Browser window (Fig. 4.33).

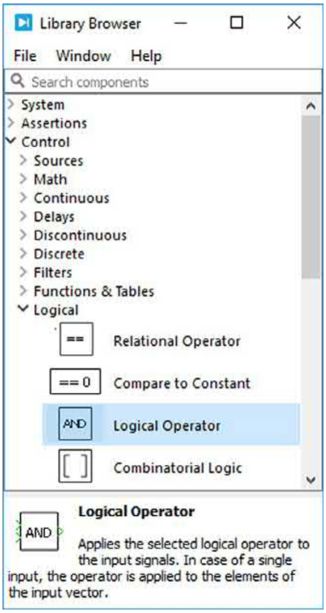

Library Browser
File
Window
Help
Search components
Delays
Discontinuous
Saturation
Dead Zone
Quantizer
Relay
Signal Switch
Manual Signal Switch
Hit Crossing
Comparator
+
-

Comparator

If the non-inverting input is greater than the inverting input, the output is 1. The output will be 0 if the non-inverting input is less than the inverting one. The output does not change if both inputs are equal

FIG. 4.33 The Comparator block.

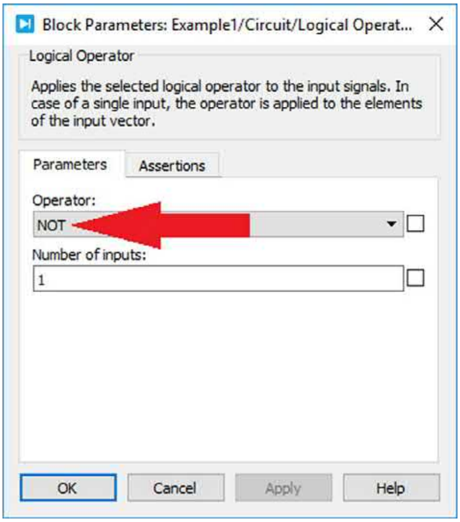
The required not gates are produced with the aid of Logical Operator block (Fig. 4.34). Set the Operator to NOT in order to obtain a not gate (Fig. 4.35).


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
==
Relational Operator
== 0
Compare to Constant
AND
Logical Operator
[ ]
Combinatorial Logic
AND

Logical Operator

Applies the selected logical operator to the input signals. In case of a single input, the operator is applied to the elements of the input vector.

FIG. 4.34 The Logical Operator block.



Block Parameters: Example1/Circuit/Logical Operat...

Logical Operator

Applies the selected logical operator to the input signals. In case of a single input, the operator is applied to the elements of the input vector.

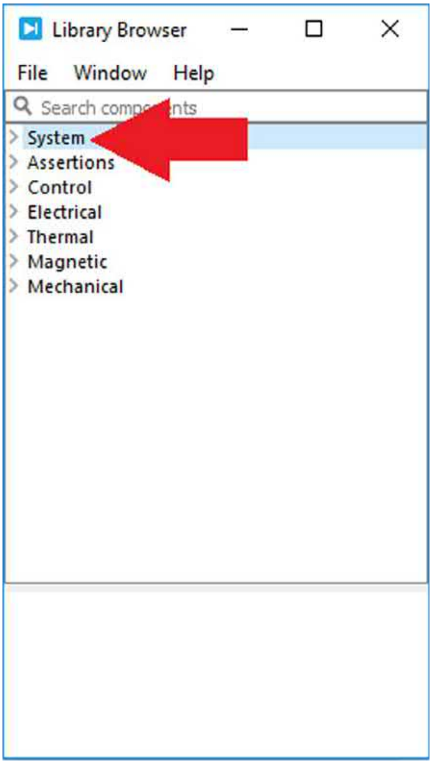
Operator: NOT

Number of inputs: 1

Parameters
Assertions
OK
Cancel
Apply
Help

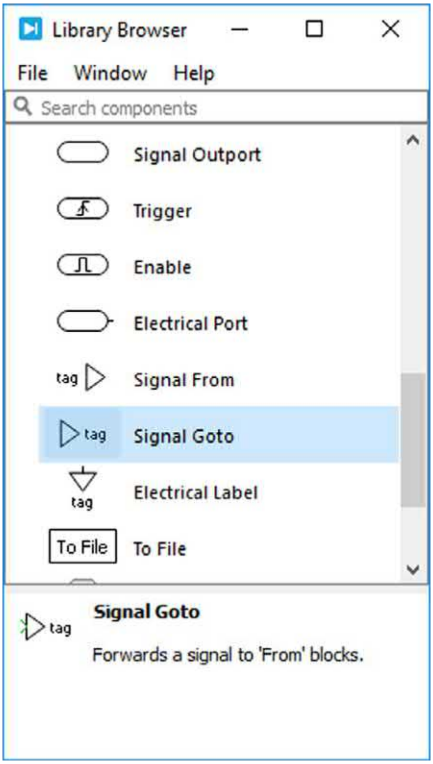
FIG. 4.35 The Logical Operator block can be turned into a not block by selecting the NOT.

The calculated signals are transferred to the gate of IGBTs with the aid of 12 Signal Goto blocks. The Signal Goto block can be found in the System section of Library Browser (Figs. 4.36 and 4.37).



Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Thermal
Magnetic
Mechanical

FIG. 4.36 The System section of Library Browser window.

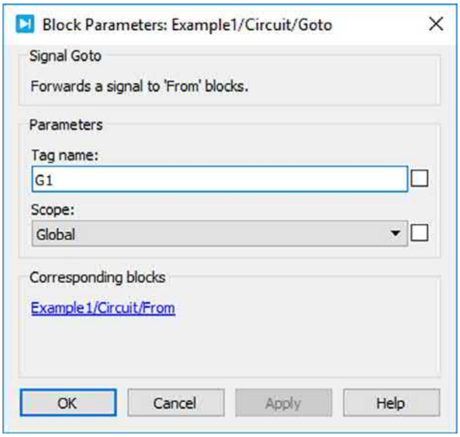

Library Browser
File Window Help
Search components
Signal Outport
Trigger
Enable
Electrical Port
tag
Signal From
tag
Signal Goto
tag
Electrical Label
To File
To File
tag

Signal Goto

Forwards a signal to 'From' blocks.

FIG. 4.37 The Signal Goto block.

The Tag name box of Signal Goto blocks is filled with G1,G2,G3,…,G12. Gi is the signal, which goes to the gate of i-th IGBT (Fig. 4.38).


Block Parameters: Example1/Circuit/Goto

Signal Goto

Forwards a signal to 'From' blocks.

Parameters

Tag name: G1 ☐

Scope: Global ☐

Corresponding blocks

Example1/Circuit/From

OK
Cancel
Apply
Help

FIG. 4.38 The Signal Goto block G1 settings. The Signal Goto blocks G2, G3, G4, …, G12 have G2, G3, G4, …, G12 in their Tag name box, respectively.

4.3.3 Simulation of circuit

Click the Simulink parameters… to set the desired simulation interval and solver (Fig. 4.39).

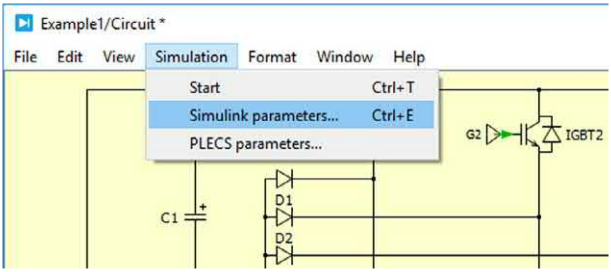
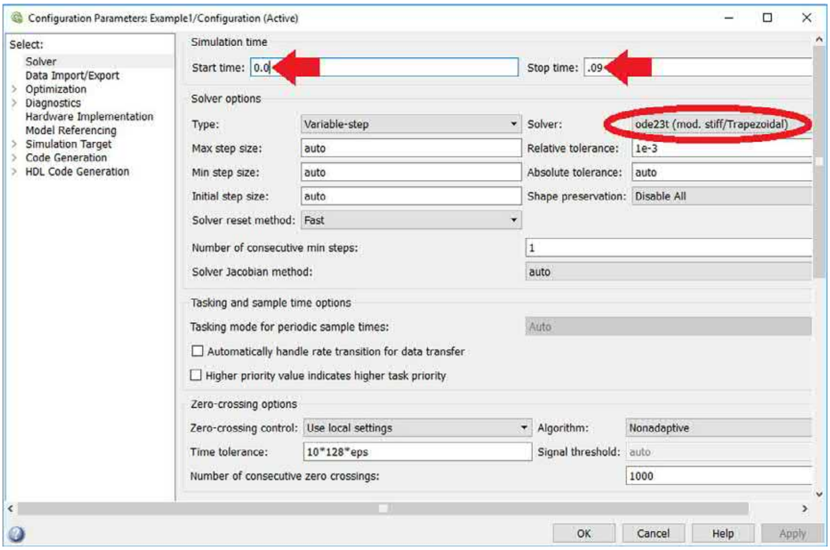


FIG. 4.39 Use the Simulink parameters to set the simulation settings.

After you clicked the Simulink parameters… the window shown in Fig. 4.40 will appear. Use the Stop time box to set the desired simulation interval. Also, use the Solver to select the desired type of solver. Selection of ode23t(mod, stiff/Trapezoidal) is suggested.


Configuration Parameters: Example1/Configuration (Active)

Select:
- Solver
- Data Import/Export
- Optimization
- Diagnostics
- Hardware Implementation
- Model Referencing
- Simulation Target
- Code Generation
- HDL Code Generation

Simulation time

Start time: 0.0
Stop time: .09

Solver options

Type: Variable-step
Solver: ode23t (mod. stiff/Trapezoidal)
Max step size: auto
Relative tolerance: 1e-3
Min step size: auto
Absolute tolerance: auto
Initial step size: auto
Shape preservation: Disable All
Solver reset method: Fast
Number of consecutive min steps: 1
Solver Jacobian method: auto

Tasking and sample time options

Tasking mode for periodic sample times: Auto
☐ Automatically handle rate transition for data transfer
☐ Higher priority value indicates higher task priority

Zero-crossing options

Zero-crossing control: Use local settings
Algorithm: Nonadaptive
Time tolerance: 10*128*eps
Signal threshold: auto
Number of consecutive zero crossings: 1000

FIG. 4.40 Configuration Parameters window.

After the desired simulation interval and Solver is entered, you can run the simulation. Click the Start in order to run the simulation (Fig. 4.41).

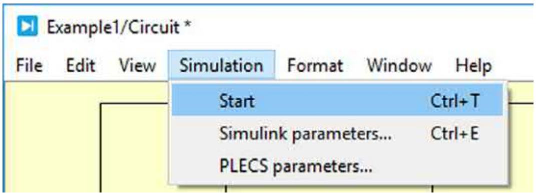

Example1/Circuit *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Simulink parameters... Ctrl+E
PLECS parameters...

FIG. 4.41 Click the Start to run the simulation.

You can run the simulation by clicking the run button in the Simulink environment as well (Fig. 4.42).

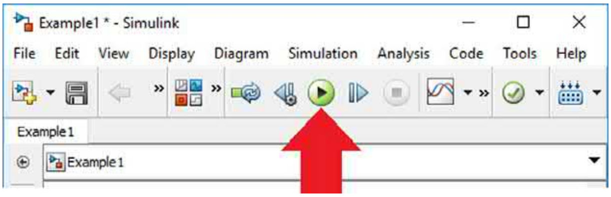

Example1 * - Simulink
File
Edit
View
Display
Diagram
Simulation
Analysis
Code
Tools
Help
Example1
Example1

FIG. 4.42 The Run button.

After the simulation is run, you can double click the scope block and see the line-line voltage waveform (Fig. 4.43).

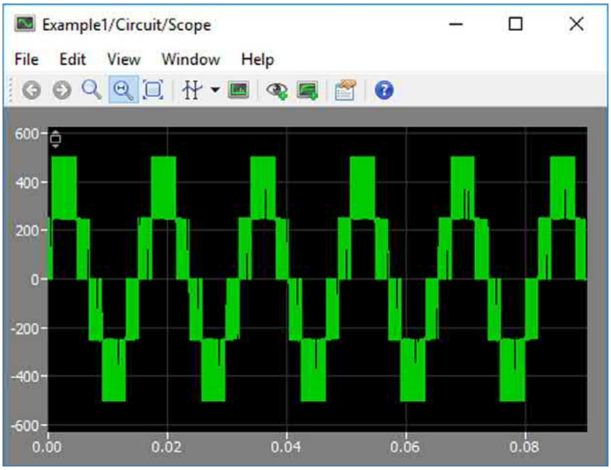
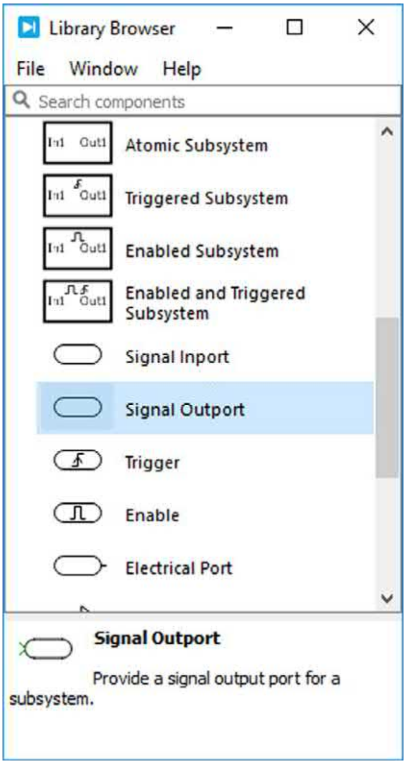


FIG. 4.43 Line-line voltage of multilevel inverter.

4.4 Sending/receiving signals to/from Simulink environment

You can send/receive the signals to/from Simulink environment easily with the aid of Signal Outport and Signal Inport blocks, respectively. The process of sending and receiving a signal is shown with two examples.

Assume that we want to send the line-line voltage to Simulink environment. In order to do this, add a Signal Outport block (Fig. 4.44) to the schematic as shown in Fig. 4.45. The Signal Outport block can be found in the System section of Library Browser (Fig. 4.46).


Library Browser
File
Window
Help
Search components
In1 Out1
Atomic Subsystem
Triggered Subsystem
Enabled Subsystem
Enabled and Triggered Subsystem
Signal Inport
Signal Outport
Trigger
Enable
Electrical Port

Signal Outport

Provide a signal output port for a subsystem.

FIG. 4.44 Signal Outport block.

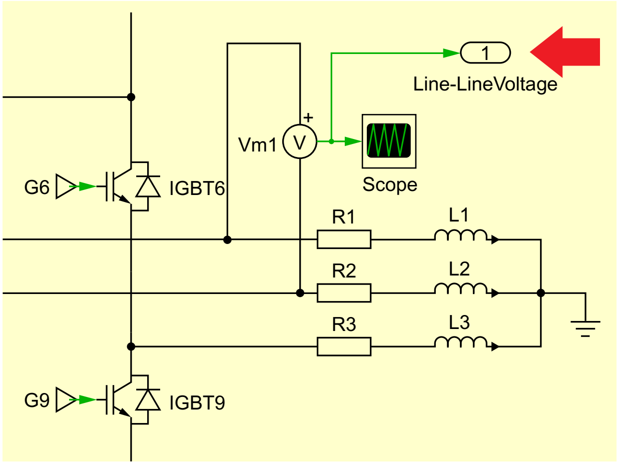

1
Line-LineVoltage
+
Vm1
V
Scope
G6
IGBT6
R1
L1
R2
L2
R3
L3
G9
IGBT9

FIG. 4.45 Addition of a Signal Outport block to the schematic. The default name of Signal Outport block is Out1. Double click the default name in order to change it to Line-LineVoltage.

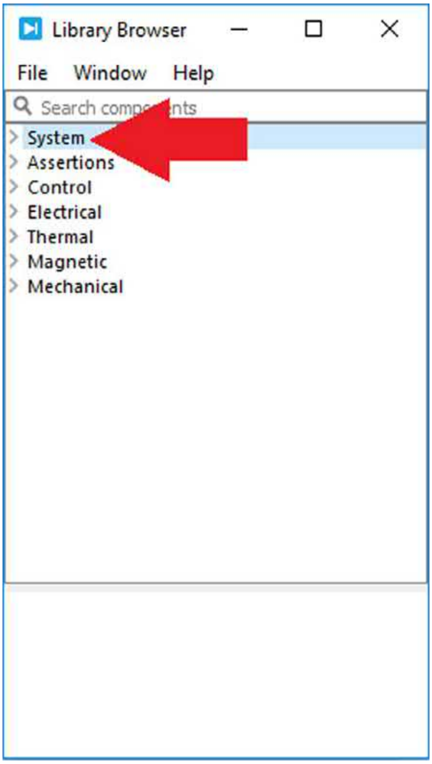

Library Browser
File Window Help
Search components
System
Assertions
Control
Electrical
Thermal
Magnetic
Mechanical

FIG. 4.46 The system section of Library Browser.

After the Signal Outport block is added to the schematic, an output port is added to the Circuit block in the Simulink environment (Fig. 4.47). You can connect the output port to oscilloscope or To Workspace blocks (Fig. 4.48).

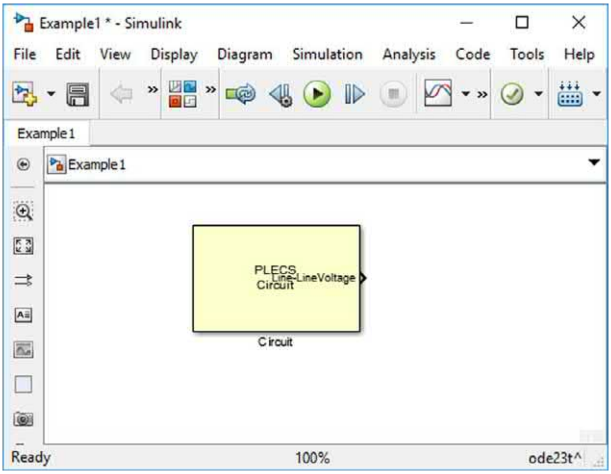

Example1 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example1
Example1
PLECS Circuit
Line-LineVoltage
Circuit
Ready
100%
ode23t

FIG. 4.47 An output is added to the PLECS Circuit block.

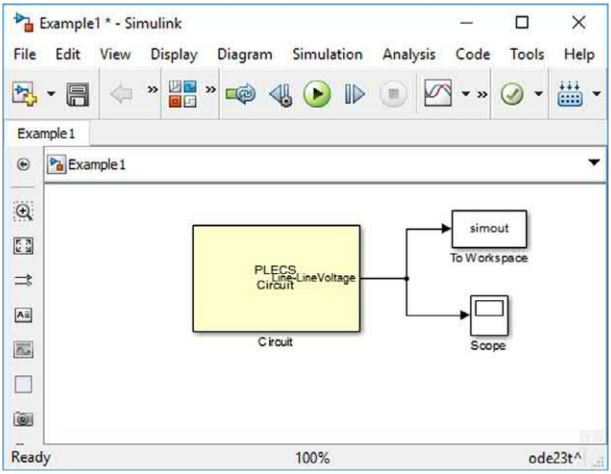

Example1 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example1
Example1
PLECS Circuit
Line-LineVoltage
Circuit
simout
To Workspace
Scope
Ready
100%
ode23t^

FIG. 4.48 Connecting the output port to other blocks.

The schematic inside the Circuit block can receive signals from the Simulink environment as well. For instance, assume we want to set the value of input DC source form Simulink environment. The independent DC source in the input of inverter is replaced with a controlled voltage source (Fig. 4.49). The controlled voltage source can be found in the Sources section of Library Browser window (Fig. 4.50).

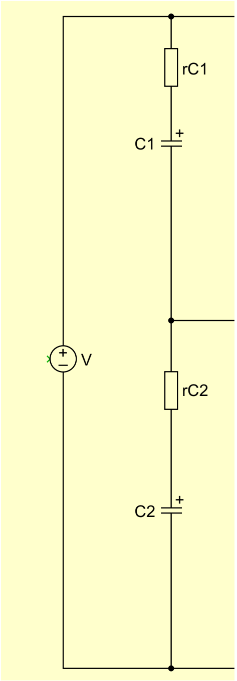
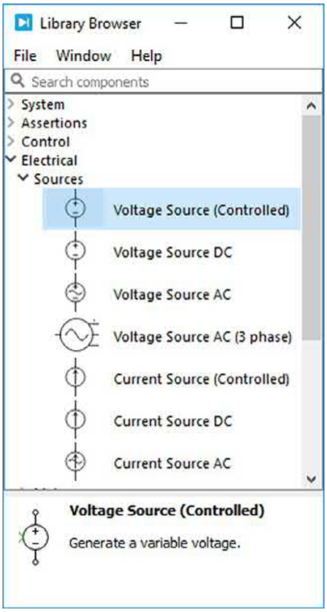

rC1
C1
+
V
rC2
C2
+

FIG. 4.49 Addition of a controlled voltage source block to the schematic.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Voltage Source (Controlled)
Voltage Source DC
Voltage Source AC
Voltage Source AC (3 phase)
Current Source (Controlled)
Current Source DC
Current Source AC

Voltage Source (Controlled)

Generate a variable voltage.

FIG. 4.50 Voltage Source (Controlled) block can be found in the Sources section of Library Browser.

The input port of the placed controlled voltage source is connected to a Signal Inport block (Fig. 4.51). The Signal Inport block (Fig. 4.52) can be found in the System section of Library Browser window (Fig. 4.53).

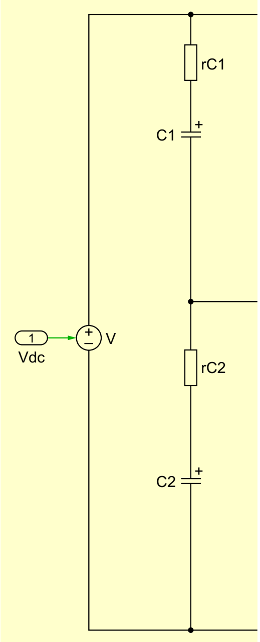
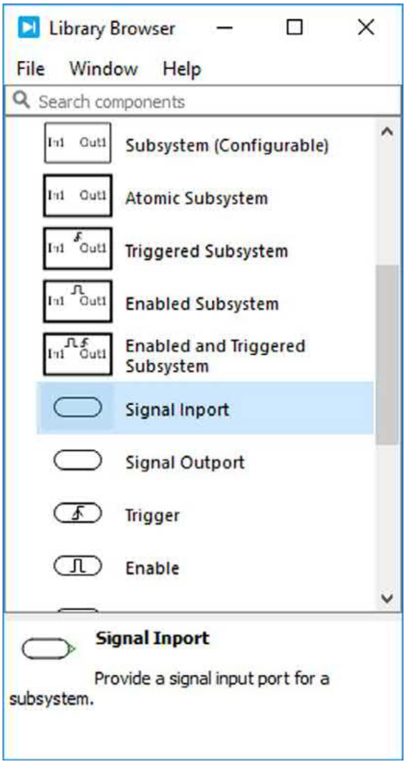

rC1
C1
+
1
Vdc
V
rC2
C2
+

FIG. 4.51 Addition of a Signal Inport block to the schematic.


Library Browser
File
Window
Help
Search components
In1 Out1
Subsystem (Configurable)
Atomic Subsystem
Triggered Subsystem
Enabled Subsystem
Enabled and Triggered Subsystem
Signal Inport
Signal Outport
Trigger
Enable

Signal Inport

Provide a signal input port for a subsystem.

FIG. 4.52 The Signal Inport block.

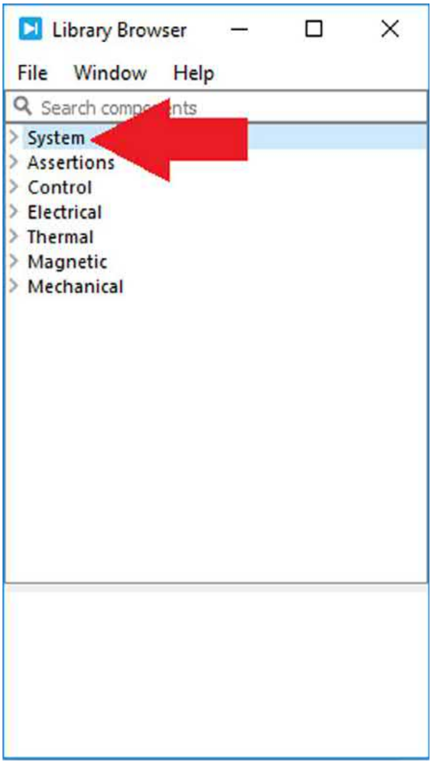

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Thermal
Magnetic
Mechanical

FIG. 4.53 The System section of Library Browser.

After you add the Signal Inport block to the schematic, an input port is added to the Circuit block inside the Simulink environment (Fig. 4.54).

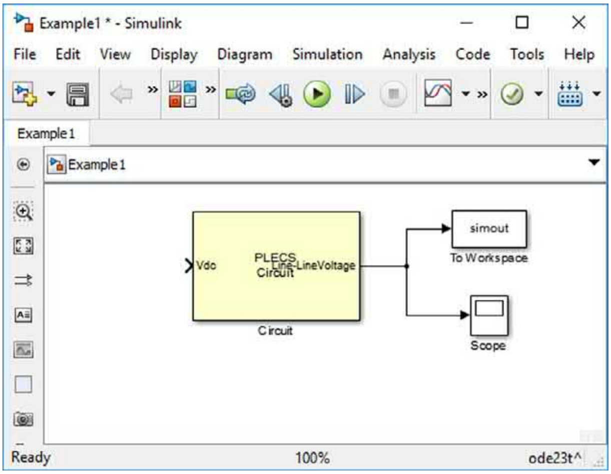

Example1 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example1
Example1
Vdc
PLECS Circuit
Line-LineVoltage
Circuit
simout
To Workspace
Scope
Ready
100%
ode23t

FIG. 4.54 An input port is added to the PLECS Circuit block.

The input port can be connected to a Constant block to set the value of input DC voltage (Fig. 4.55).

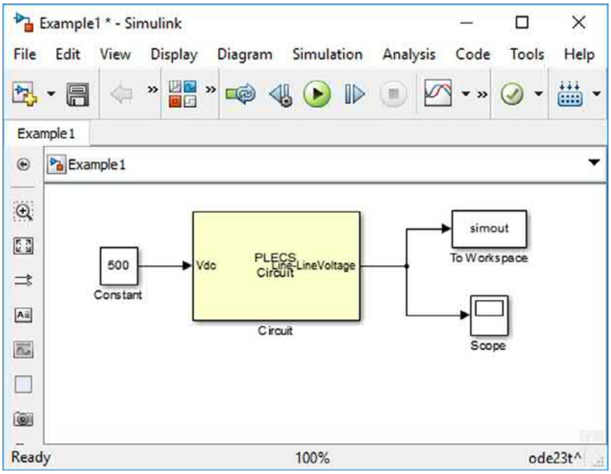

Example1 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example1
Example1
500
Constant
Vdc
PLECS Circuit
Line-LineVoltage
Circuit
simout
To Workspace
Scope
Ready
100%
ode23t

FIG. 4.55 The input port is connected to a constant block.

4.5 Simulation of a cascaded inverter

A single-phase inverter is shown in Fig. 4.56.

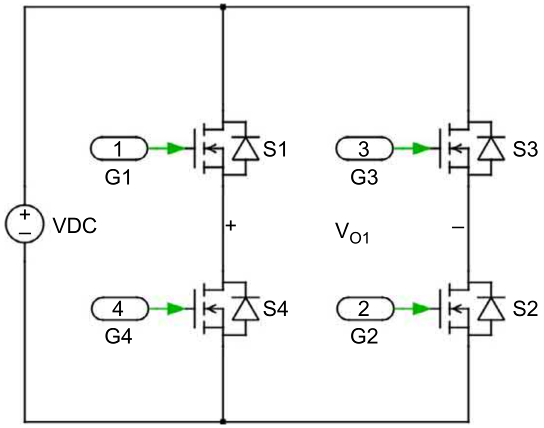

1
G1
S1
3
G3
S3
+
−
VDC
+
V01
−
4
G4
S4
2
G2
S2

FIG. 4.56 Single-phase inverter circuit.

This topology can produce three different output voltages (V_{O1}). The output voltage can be -VDC, 0, +VDC. When switches S3 and S4 are closed, the output voltage is –VDC. When switches S1 and S2 are closed, the output voltage is +VDC. When S1 and S3 are closed or S2 and S4 are closed, the output voltage will be 0 volt. In order to obtain the output waveform shown in part (a) of Fig. 4.57, the switching pattern shown in part (b) of Fig. 4.57 must be applied to the inverter.

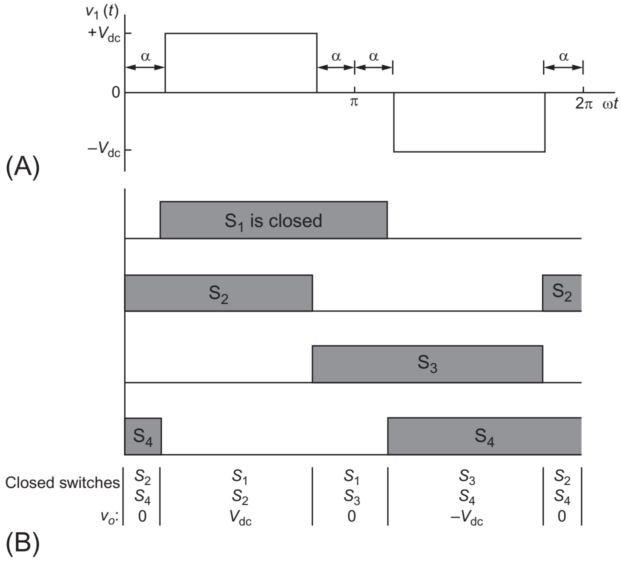

v1 (t)
+Vdc
0
−Vdc
α
α
α
α
π
2π ωt
(A)
S1 is closed
S2
S2
S3
S4
S4
Closed switches
vo:
S2 S4 0
S1 S2 Vdc
S1 S3 0
S3 S4 −Vdc
S2 S4 0
(B)

FIG. 4.57 (A) Output voltage, (B) Required switching pattern to produce waveform shown in part (A).

The building block shown in Fig. 4.56 can be used to make a larger inverter. Circuit shown in Fig. 4.58 uses cascaded connection of two single-phase inverters. This circuit can produces -2VDC, -VDC, 0, +VDC, and +2VDC in its outputs. Increasing the levels of output voltage decreases the Total Harmonic Distortion (THD) of output voltage.

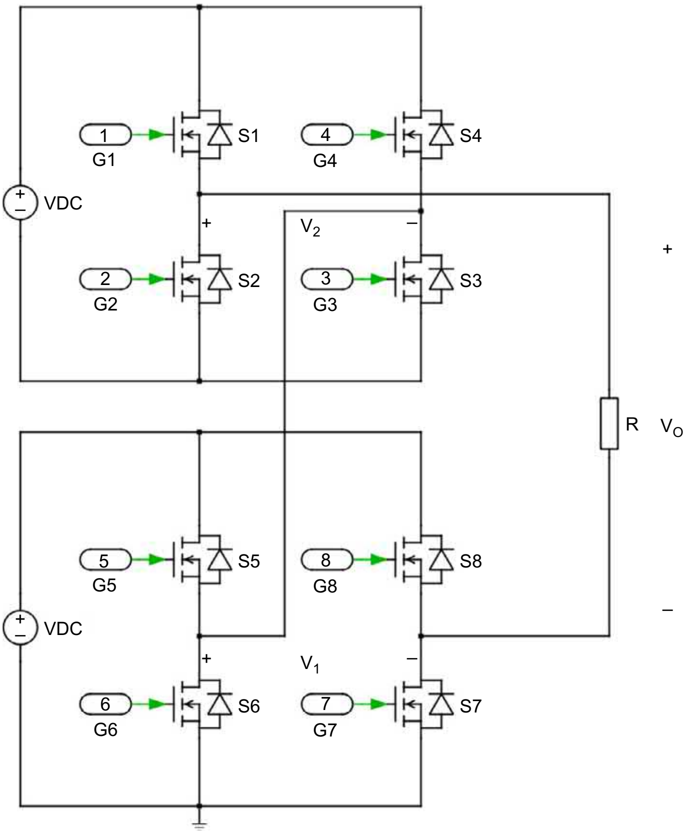

1
G1
S1
4
G4
S4
VDC
+
V_2
−
2
G2
S2
3
G3
S3
+
R
V_O
5
G5
S5
8
G8
S8
VDC
−
+
V_1
−
6
G6
S6
7
G7
S7

FIG. 4.58 Cascade connection of two single-phase inverter. Note that $v_O = v_1 + v_2$.

Fig. 4.59 shows the output voltage of circuit shown in Fig. 4.58 and the required switching pattern.

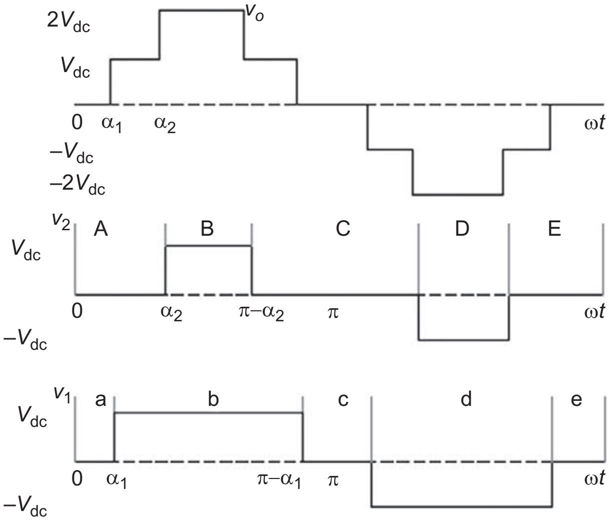


FIG. 4.59 Voltage waveforms of cascaded inverter (Fig. 4.58). Note that $v_O = v_1 + v_2$.

Tables 4.1 and 4.2 show the status of switches in order to produce the output voltage shown in Fig. 4.59. 1 shows a closed switch, i.e., short circuit, and 0 shows the opened switch, i.e., open circuit.

TABLE 4.1 Control signals for upper inverter switches (see Figs. 4.58 and 4.59).

Interval name	Description of interval	S1 status	S2 status	S3 status	S4 status
A	$0 < \omega t < \alpha_2$	1	0	0	1
B	$\alpha_2 < \omega t < \pi - \alpha_2$	1	0	1	0
C	$\pi - \alpha_2 < \omega t < \pi + \alpha_2$	1	0	0	1
D	$\pi + \alpha_2 < \omega t < 2\pi - \alpha_2$	0	1	0	1
E	$2\pi - \alpha_2 < \omega t < 2\pi$	1	0	0	1

TABLE 4.2 Control signals for lower inverter switches (see Figs. 4.58 and 4.59).

Interval name	Description of Interval	S5 status	S6 status	S7 status	S8 status
a	$0<\omega t<\alpha_1$	1	0	0	1
b	$\alpha_1<\omega t<\pi-\alpha_1$	1	0	1	0
c	$\pi-\alpha_1<\omega t<\pi+\alpha_1$	1	0	0	1
d	$\pi+\alpha_1<\omega t<2\pi-\alpha_1$	0	1	0	1
e	$2\pi-\alpha_1<\omega t<2\pi$	1	0	0	1

We want to simulate the circuit shown in Fig. 4.58 for $\alpha_1=20°$ and $\alpha_2=40°$. We use the Simulink model shown in Fig. 4.60.

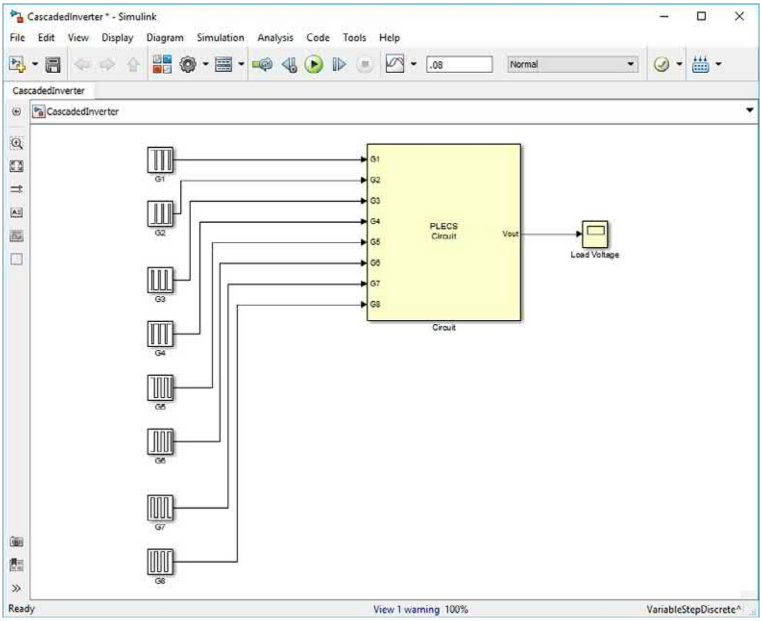

CascadedInverter * - Simulink
File
Edit
View
Display
Diagram
Simulation
Analysis
Code
Tools
Help
.08
Normal
CascadedInverter
CascadedInverter
G1
G2
G3
G4
G5
G6
G7
G8
G1
G2
G3
G4
G5
G6
G7
G8
PLECS
Circuit
Vout
Circuit
Load Voltage
Ready
View 1 warning
100%
VariableStepDiscrete

FIG. 4.60 Simulink model of cascaded inverter.

The schematic of circuit inside the PLECS Circuit block is shown in Fig. 4.61.

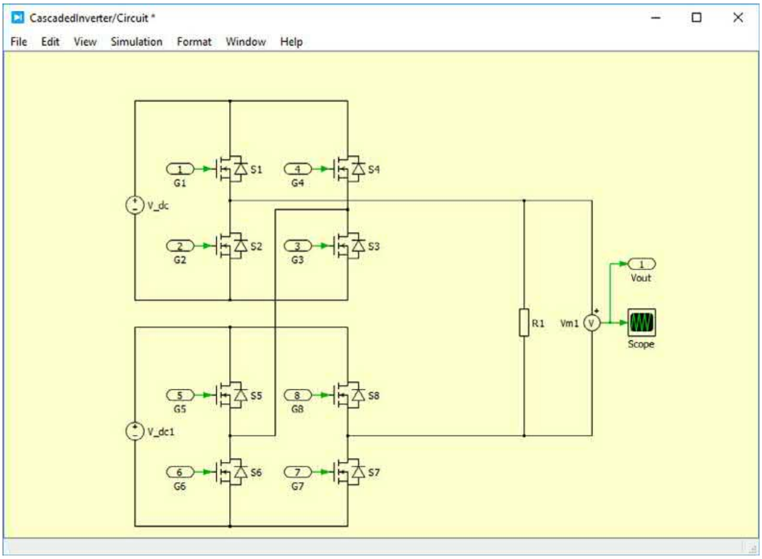
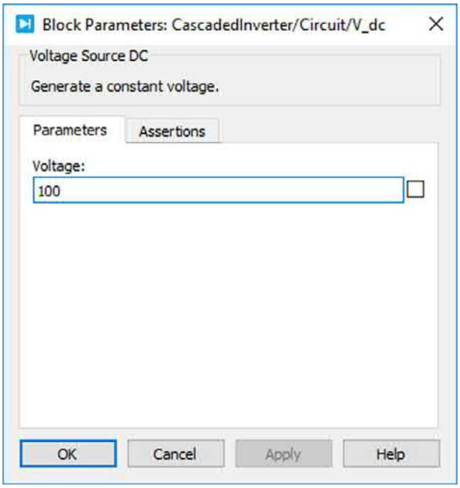

CascadedInverter/Circuit *
File Edit View Simulation Format Window Help
G1
S1
G4
S4
V_dc
G2
S2
G3
S3
Vout
R1
Vm1
V
Scope
G5
S5
G8
S8
V_dc1
G6
S6
G7
S7

FIG. 4.61 Schematic of cascaded inverter.

The settings of used components are shown in Figs. 4.62–4.64.

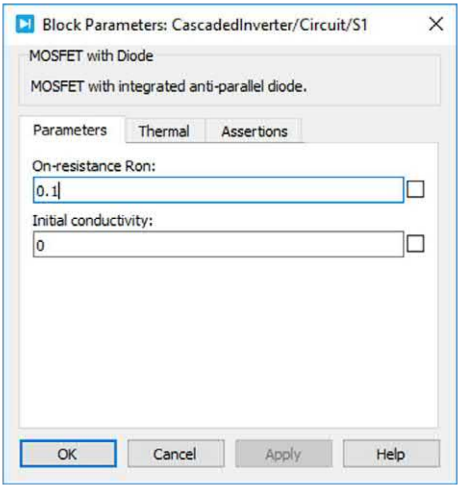

Block Parameters: CascadedInverter/Circuit/V_dc
Parameters
Assertions
OK
Cancel
Apply
Help

Voltage Source DC

Generate a constant voltage.

Voltage: 100

FIG. 4.62 Setting of V_dc block. The V_dc1 block has the same settings.



Block Parameters: CascadedInverter/Circuit/S1

MOSFET with Diode

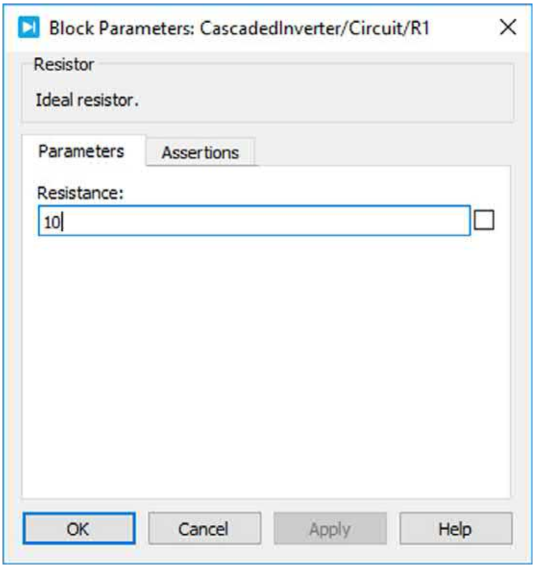
MOSFET with integrated anti-parallel diode.

On-resistance Ron: 0.1 ☐

Initial conductivity: 0 ☐

Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

FIG. 4.63 The MOSFET S1 settings. S2, S3, S4, …, S8 have the same settings.



Block Parameters: CascadedInverter/Circuit/R1

Resistor

Ideal resistor.

Resistance: 10 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.64 Resistor R1 settings.

We use the Repeating Sequence block (Fig. 4.65) to produce the required gate signals. The MOSFET will be closed once we send 1 to its gate and will be opened once we send 0 to its gate.

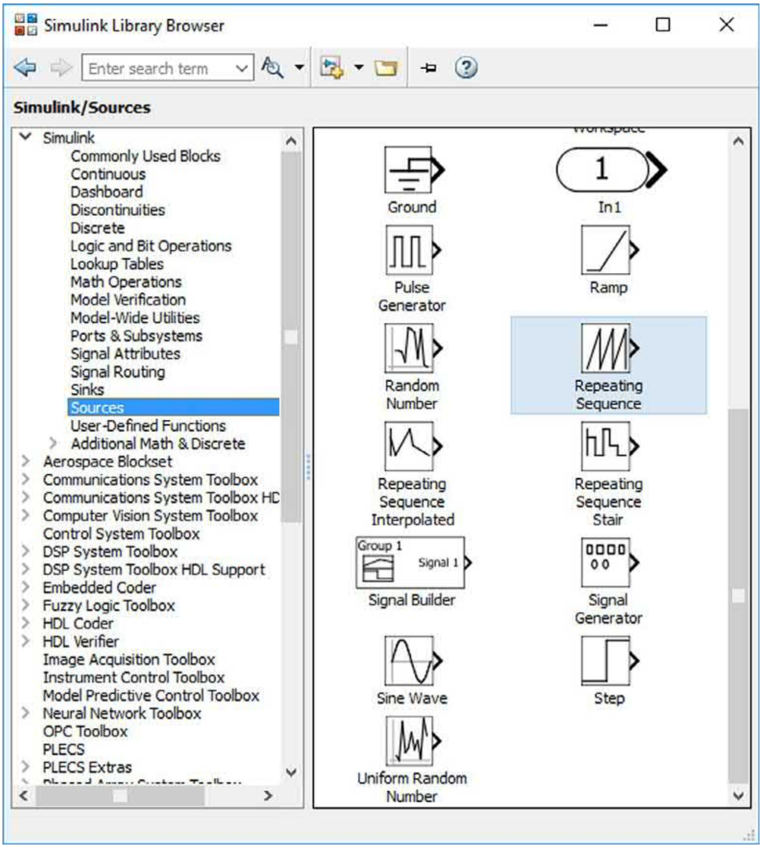

Simulink Library Browser
Enter search term
Simulink/Sources
Simulink
Commonly Used Blocks
Continuous
Dashboard
Discontinuities
Discrete
Logic and Bit Operations
Lookup Tables
Math Operations
Model Verification
Model-Wide Utilities
Ports & Subsystems
Signal Attributes
Signal Routing
Sinks
Sources
User-Defined Functions
Additional Math & Discrete
Aerospace Blockset
Communications System Toolbox
Communications System Toolbox HD
Computer Vision System Toolbox
Control System Toolbox
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Ground
1
In1
Pulse Generator
Ramp
Random Number
Repeating Sequence
Repeating Sequence Interpolated
Repeating Sequence Stair
Group 1
Signal 1
Signal Builder
Signal Generator
Sine Wave
Step
Uniform Random Number

FIG. 4.65 Repeating Sequence block.

Assume that you want to produce the waveform shown in Fig. 4.66 with the aid of Repeating Sequence block. The given graph is given in terms of angles. We need to know the switching times when working with the Repeating Sequence block.

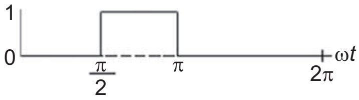

1
0
$\frac{\pi}{2}$
π
2π
ωt

FIG. 4.66 Sample waveform. The vertical axis is angle in radyan.

We must translate the ωt axes from angles into time. We must know the period of given waveform to do the translation. Assume that the waveform shown in Fig. 4.66 has the frequency of 1 kHz (period equals to $\frac{1}{1000} = 1\,\text{ms}$).

The switching time will be $\frac{\frac{\pi}{2}}{2\pi}\times 1\,\text{ms}=0.25\,\text{ms}$ and $\frac{\pi}{2\pi}\times 1\,\text{ms}=0.5\,\text{ms}$. Fig. 4.67 shows the time domain equivalent of Fig. 4.66.

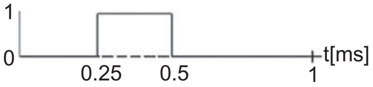
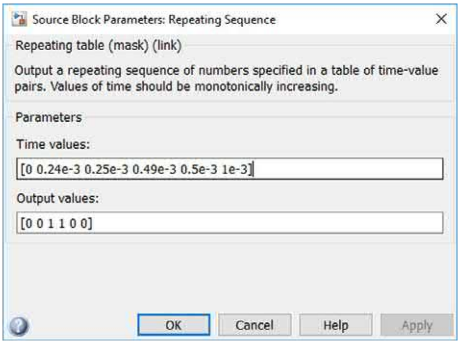


FIG. 4.67 Time domain equivalent of waveform shown in Fig. 4.66. The vertical axis is angle in seconds.

According to Fig. 4.67, the signal is 0 for $0<t<0.25\,\text{ms}$, 1 for $0.25\,\text{ms}<t<0.5\,\text{ms}$, and 0 for $0.5\,\text{ms}<t<1\,\text{ms}$. Assume the transition from 0 to 1 or 1 to 0 takes 0.01 ms. Then, the signal is 0 for [0,0.25 ms − 0.01 ms], the transition interval is [0.25 ms − 0.01 ms, 0.25 ms − 0.01 ms + 0.01 ms]. After the transition, the signal will be at 1 and remain there up to next transition, i.e., the signal is 1 for [0.25 ms, 0.5 ms − 0.01 ms]. There is another transition from 1 to 0 at [0.5 ms − 0.01 ms, 0.5 ms]. After that transition the signal will stay at 0, i.e., the signal is 0 for [0.5 ms, 1 ms].

Fig. 4.68 shows the required setting for Repeating Sequence block in order to produce the aforementioned signal.


Source Block Parameters: Repeating Sequence

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

Time values:

[0 0.24e-3 0.25e-3 0.49e-3 0.5e-3 1e-3]

Output values:

[0 0 1 1 0 0]

FIG. 4.68 Required setting to produce the waveform shown in Fig. 4.67. It is assumed that transitions take 0.01s.

The output signal of Repeating Sequence block shown in Fig. 4.68 is shown in Fig. 4.69.

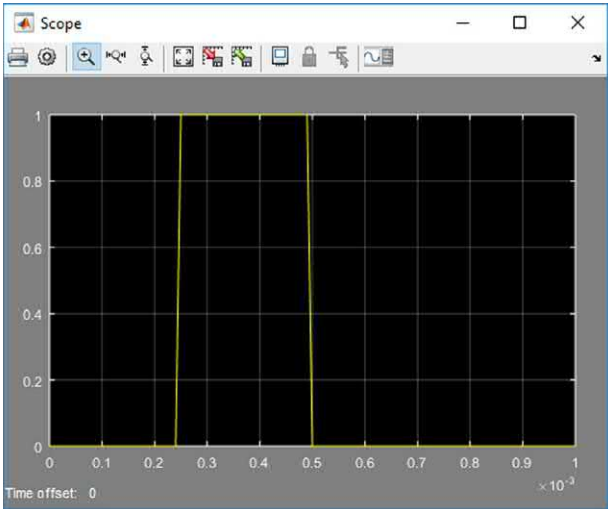
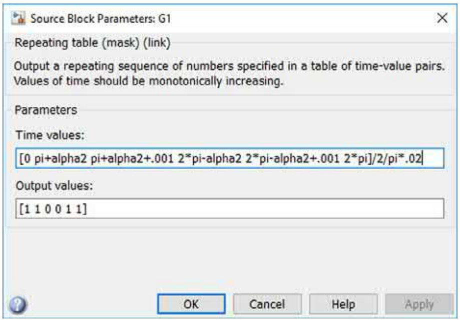


FIG. 4.69 Output of Repeating Sequence block with the setting shown in Fig. 4.68. Repeating Sequence block produces a periodic output. Only one period is shown in this figure.

The required setting for blocks G1, G2, G3, …, G8 is shown in Figs. 4.70–4.77. The setting for this block is done with respect to Table 4.1 and 4.2.


Source Block Parameters: G1

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

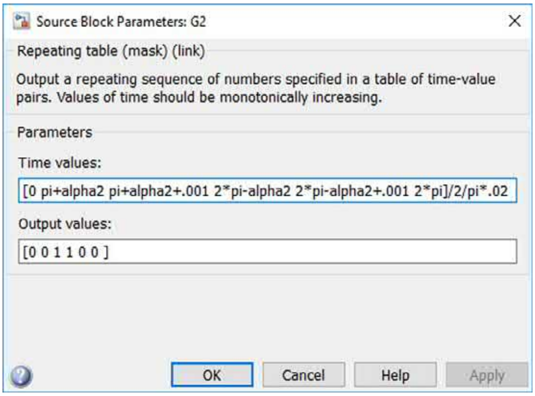
Time values:

`[0 pi+alpha2 pi+alpha2+.001 2*pi-alpha2 2*pi-alpha2+.001 2*pi]/2/pi*.02`

Output values:

`[1 1 0 0 1 1]`

FIG. 4.70 Settings of block G1. It is assumed that transitions takes at $\frac{0.001}{2\pi} \times 0.02 = 3.2\,\mu\text{s}$.


Source Block Parameters: G2

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

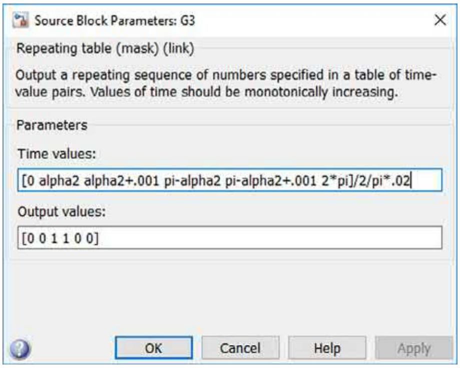
Time values:

`[0 pi+alpha2 pi+alpha2+.001 2*pi-alpha2 2*pi-alpha2+.001 2*pi]/2/pi*.02`

Output values:

`[0 0 1 1 0 0 ]`

FIG. 4.71 Settings of block G2. It is assumed that transitions takes at $\frac{0.001}{2\pi} \times 0.02 = 3.2\,\mu\text{s}$.


Source Block Parameters: G3

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

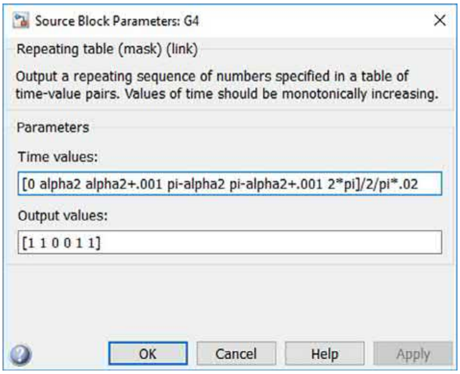
Time values:

`[0 alpha2 alpha2+.001 pi-alpha2 pi-alpha2+.001 2*pi]/2/pi*.02`

Output values:

`[0 0 1 1 0 0]`

FIG. 4.72 Settings of block G3. It is assumed that transitions takes at $\frac{0.001}{2\pi} \times 0.02 = 3.2\,\mu\text{s}$.


Source Block Parameters: G4

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

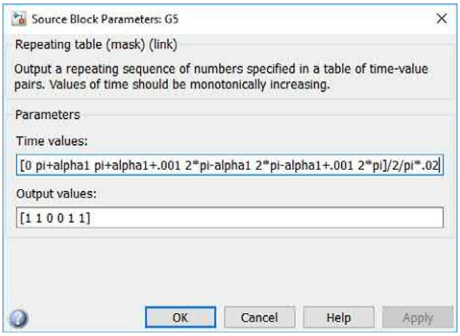
Time values:

`[0 alpha2 alpha2+.001 pi-alpha2 pi-alpha2+.001 2*pi]/2/pi*.02`

Output values:

`[1 1 0 0 1 1]`

FIG. 4.73 Settings of block G4. It is assumed that transitions takes at $\frac{0.001}{2\pi} \times 0.02 = 3.2\,\mu\text{s}$.


Source Block Parameters: G5

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

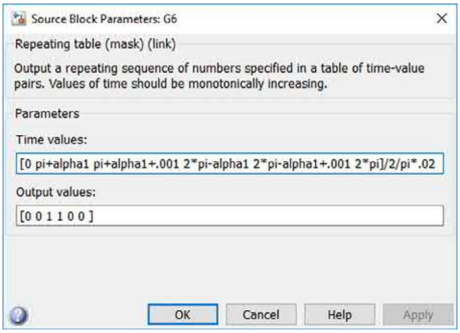
Time values:

`[0 pi+alpha1 pi+alpha1+.001 2*pi-alpha1 2*pi-alpha1+.001 2*pi]/2/pi*.02`

Output values:

`[1 1 0 0 1 1]`

FIG. 4.74 Settings of block G5. It is assumed that transitions takes at $\frac{0.001}{2\pi} \times 0.02 = 3.2\,\mu\text{s}$.


Source Block Parameters: G6

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

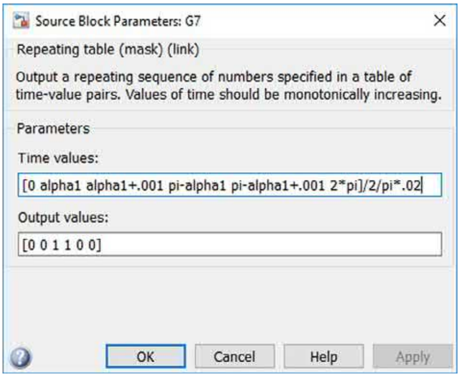
Time values:

`[0 pi+alpha1 pi+alpha1+.001 2*pi-alpha1 2*pi-alpha1+.001 2*pi]/2/pi*.02`

Output values:

`[0 0 1 1 0 0 ]`

FIG. 4.75 Settings of block G6. It is assumed that transitions takes at $\frac{0.001}{2\pi} \times 0.02 = 3.2\,\mu\text{s}$.


Source Block Parameters: G7

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

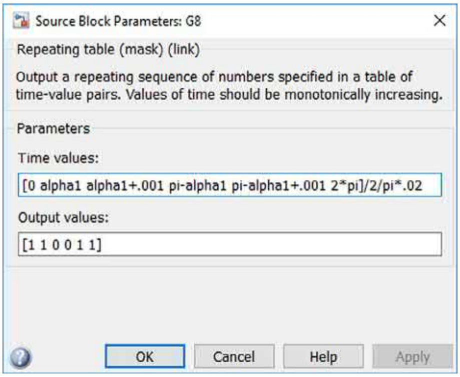
Time values:

`[0 alpha1 alpha1+.001 pi-alpha1 pi-alpha1+.001 2*pi]/2/pi*.02`

Output values:

`[0 0 1 1 0 0]`

FIG. 4.76 Settings of block G7. It is assumed that transitions takes at $\frac{0.001}{2\pi} \times 0.02 = 3.2\,\mu\text{s}$.


Source Block Parameters: G8

Repeating table (mask) (link)

Output a repeating sequence of numbers specified in a table of time-value pairs. Values of time should be monotonically increasing.

Parameters

Time values:

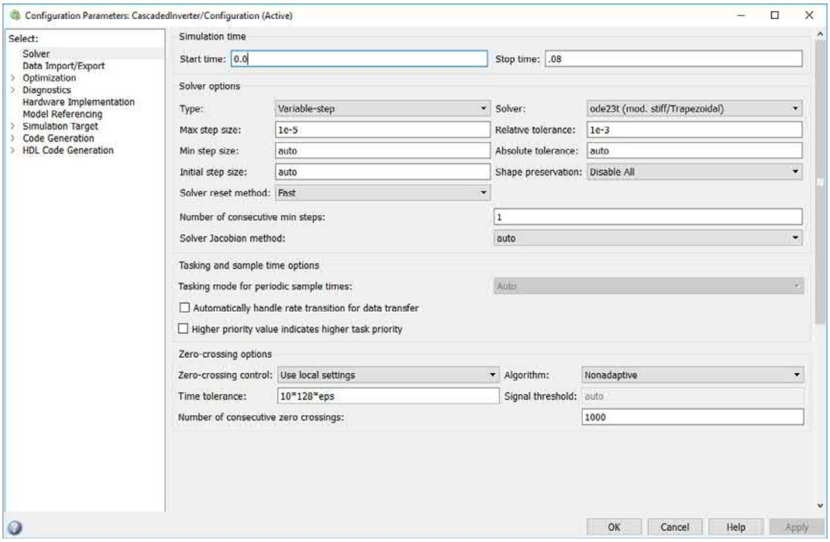
`[0 alpha1 alpha1+.001 pi-alpha1 pi-alpha1+.001 2*pi]/2/pi*.02`

Output values:

`[1 1 0 0 1 1]`

FIG. 4.77 Settings of block G8. It is assumed that transitions takes at $\frac{0.001}{2\pi} \times 0.02 = 3.2\,\mu\text{s}$.

Click the gear icon in the Simulink environment and set the solver and stop time as shown in Fig. 4.78.



Configuration Parameters: CascadedInverter/Configuration (Active)

Select:
- Solver
- Data Import/Export
- Optimization
- Diagnostics
- Hardware Implementation
- Model Referencing
- Simulation Target
- Code Generation
- HDL Code Generation

Simulation time

Start time: 0.0
Stop time: .08

Solver options

Type: Variable-step
Solver: ode23t (mod. stiff/Trapezoidal)
Max step size: 1e-5
Relative tolerance: 1e-3
Min step size: auto
Absolute tolerance: auto
Initial step size: auto
Shape preservation: Disable All
Solver reset method: Fast
Number of consecutive min steps: 1
Solver Jacobian method: auto

Tasking and sample time options

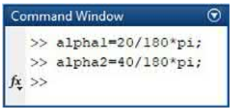
Tasking mode for periodic sample times: Auto
☐ Automatically handle rate transition for data transfer
☐ Higher priority value indicates higher task priority

Zero-crossing options

Zero-crossing control: Use local settings
Algorithm: Nonadaptive
Time tolerance: 10*128*eps
Signal threshold: auto
Number of consecutive zero crossings: 1000

FIG. 4.78 Configuration parameters window.

Don't forget to initialize the values of α_1 and α_2 before running the simulation (Fig. 4.79).



```
>> alpha1=20/180*pi;
>> alpha2=40/180*pi;
>>
```

FIG. 4.79 Initialization of variables `alpha1` and `alpha2` in radyan.

Run the simulation. The simulation result is shown in Fig. 4.80.

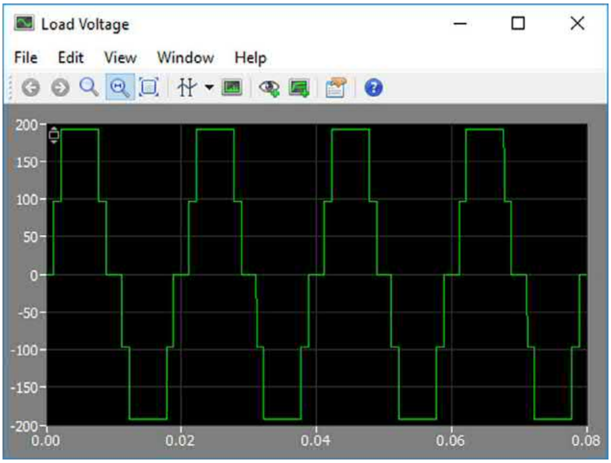


FIG. 4.80 Simulation result (load voltage).

There is no limitation on the number of cascaded stages. For instance, the inverter shown in Fig. 4.81 is composed of five stages. The produced output voltage is shown in Fig. 4.82. Note that cascaded inverters require separate DC sources, which is one of their disadvantages.

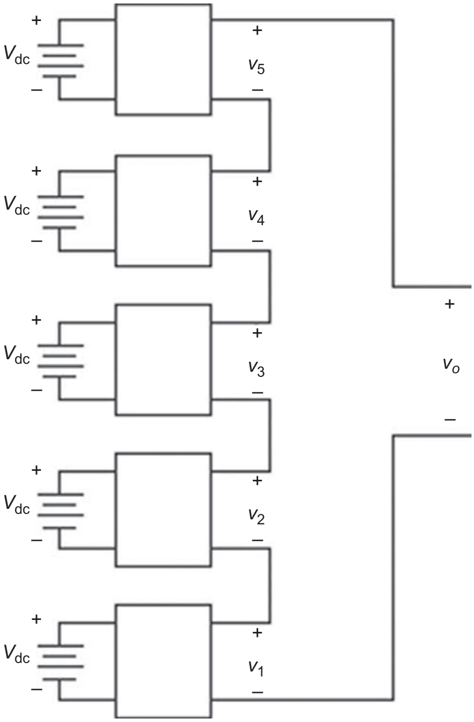


FIG. 4.81 A cascaded inverter composed of five stages.

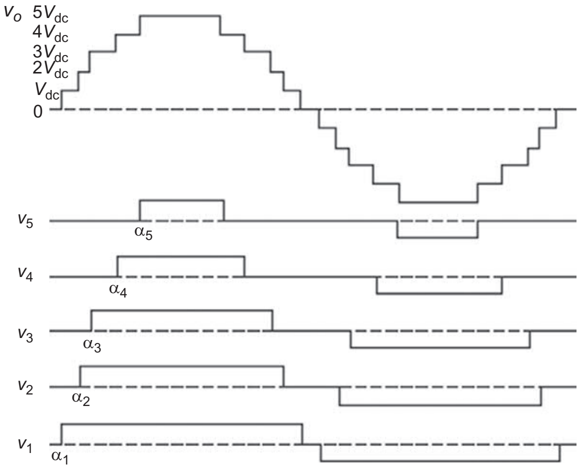


FIG. 4.82 Output voltage of inverter shown in Fig. 4.81.

4.6 Measurement with the probe block

You can use the Probe block to measure the desired quantities. Consider the simple RLC circuit shown in Fig. 4.83.

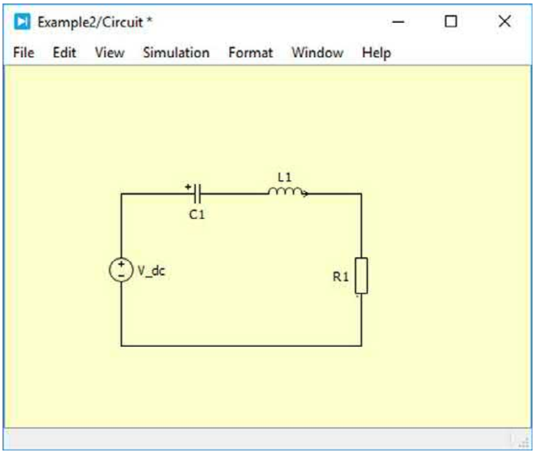

Example2/Circuit *
File Edit View Simulation Format Window Help
L1
C1
V_dc
R1

FIG. 4.83 A simple RLC circuit. V_dc = 100 V, C1 = 100 μf, L1 = 1 mH, and R = 1 Ω.

Add a PLECS Probe block to Simulink simulation (Fig. 4.84). The PLECS Probe block can be found in the PLECS section of Simulink Library Browser (Fig. 4.85)

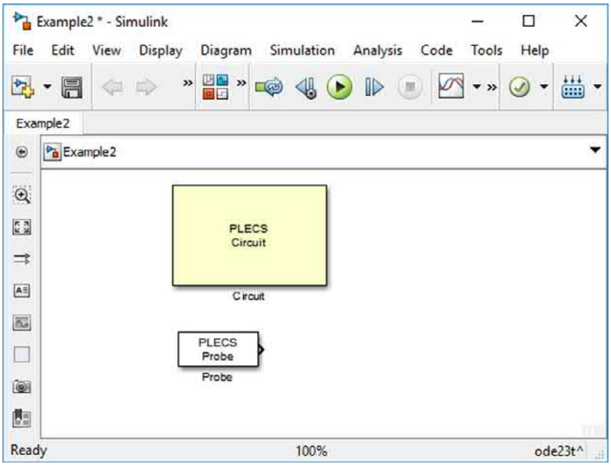

Example2 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example2
Example2
PLECS Circuit
Circuit
PLECS Probe
Probe
Ready
100%
ode23t

FIG. 4.84 Addition of a PLECS Probe block to the schematic.

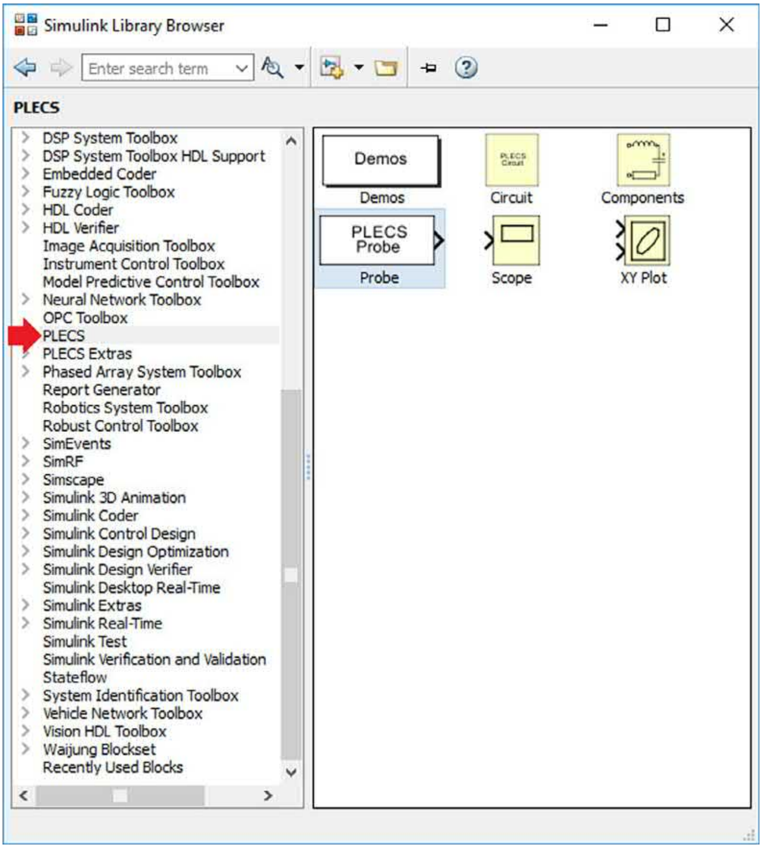

Simulink Library Browser
Enter search term
PLECS
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Desktop Real-Time
Simulink Extras
Simulink Real-Time
Simulink Test
Simulink Verification and Validation
Stateflow
System Identification Toolbox
Vehicle Network Toolbox
Vision HDL Toolbox
Waijung Blockset
Recently Used Blocks
Demos
Demos
PLECS Circuit
Circuit
Components
PLECS Probe
Probe
Scope
XY Plot

FIG. 4.85 PLECS Probe block can be found in the PLECS section of Simulink Library Browser.

Open the PLECS Probe block by double clicking on it. Open the Circuit block as well (Fig. 4.86).

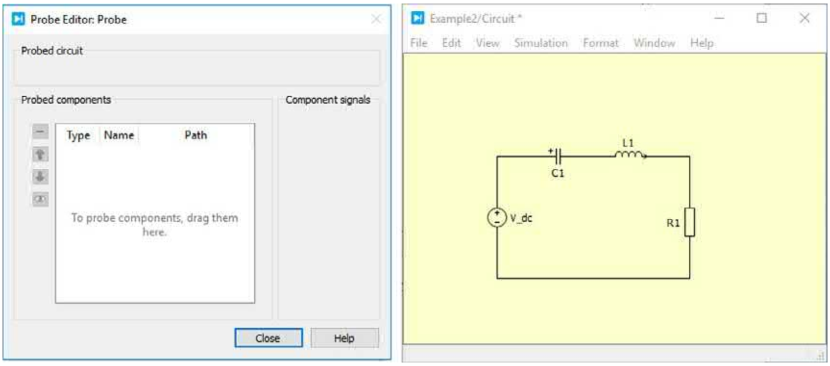
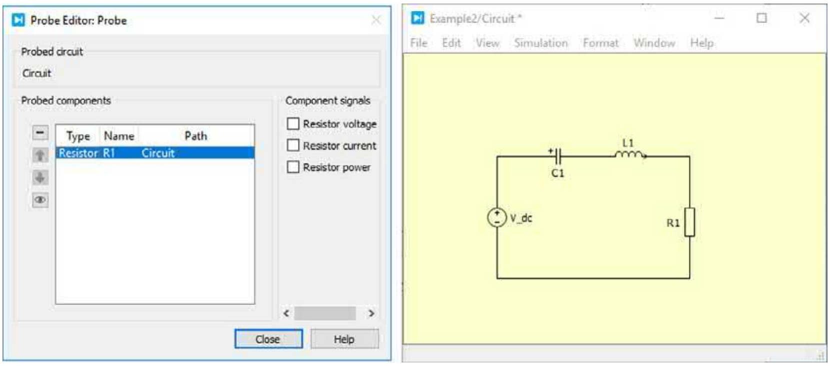

Probe Editor: Probe
Probed circuit
Probed components
Component signals
Type
Name
Path
To probe components, drag them here.
Close
Help
Example2/Circuit *
File
Edit
View
Simulation
Format
Window
Help
L1
C1
V_dc
R1

FIG. 4.86 The opened Probe block and drawn schematic.

We want to see the instantaneous dissipated power of resistor. The power can be calculated by product of current through the resistor and voltage across the resistor. Drag the resistor from the schematic and drop it in the white area of Probed components (Fig. 4.87).


Probe Editor: Probe
Probed circuit
Circuit
Probed components
Component signals
Close
Help
Example2/Circuit *
File
Edit
View
Simulation
Format
Window
Help
C1
L1
V_dc
R1

Probed components:

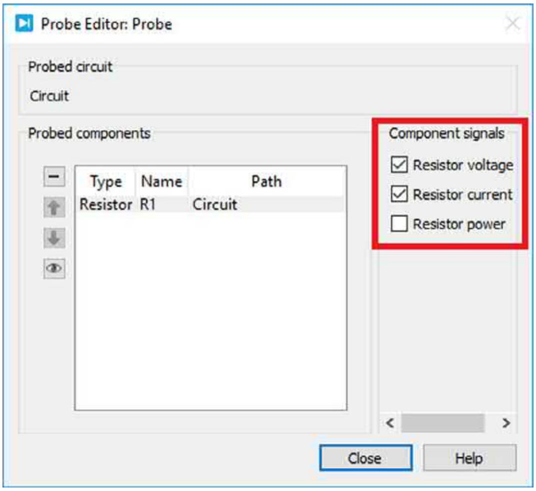
Type	Name	Path
Resistor	R1	Circuit

Component signals:

- ☐ Resistor voltage
- ☐ Resistor current
- ☐ Resistor power

FIG. 4.87 Drag and drop the resistor R1 onto the Probed components box of opened Probe block.

Select the desired quantities to be measured from the Component signals section of the window. In this case we need the voltage and current of resistor, so Resistor voltage and Resistor current of boxes are checked (Fig. 4.88).


Probe Editor: Probe
Probed circuit
Circuit
Probed components
Component signals
Close
Help

Probed components:

Type	Name	Path
Resistor	R1	Circuit

Component signals:

- ☑ Resistor voltage
- ☑ Resistor current
- ☐ Resistor power

FIG. 4.88 The measurable signals.

There is a simple dot behind the resistor symbol in the schematic (Fig. 4.89). The PLECS Probe assumes the current, which enters the dot as positive. The PLECS Probe measures the resistor voltage with respect to undotted pin, i.e., the PLECS Probe measures $v_{dotted\ pin} - v_{undotted\ pin}$.

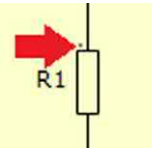


FIG. 4.89 The dot behind the resistor icon.

Add a demultiplexer block to output of PLECS Probe block. The multiplexer block separates the voltage and current. Use a multiply block to calculate the power (Fig. 4.90).

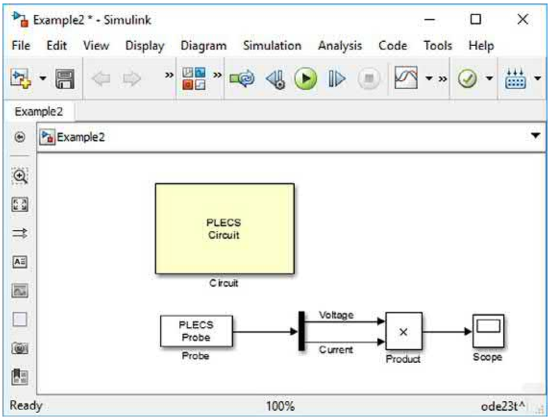

Example2 * - Simulink
File
Edit
View
Display
Diagram
Simulation
Analysis
Code
Tools
Help
Example2
Example2
PLECS Circuit
Circuit
PLECS Probe
Probe
Voltage
Current
×
Product
Scope
Ready
100%
ode23tb

FIG. 4.90 Completed Simulink model.

Simulation result is shown in Fig. 4.91.

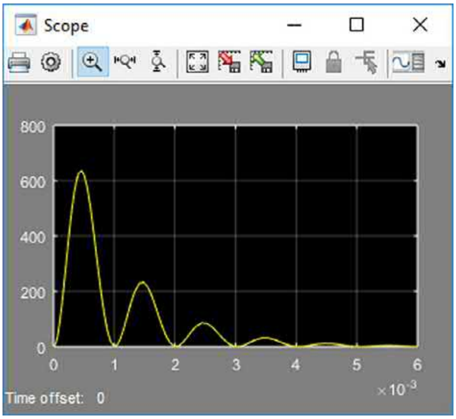


FIG. 4.91 Simulation result.

4.7 Extraction of frequency response of DC-DC converters

You can obtain the small signal-frequency response of DC-DC converters with the aid of Impulse Response Analysis block. We show how to extract the frequency response of control-to-output transfer function in this section.

Assume a buck converter like the one shown in Fig. 4.92. Values of components can be read from the schematic.

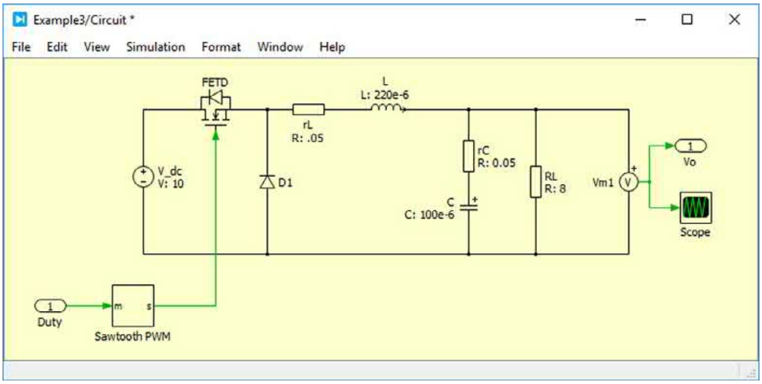
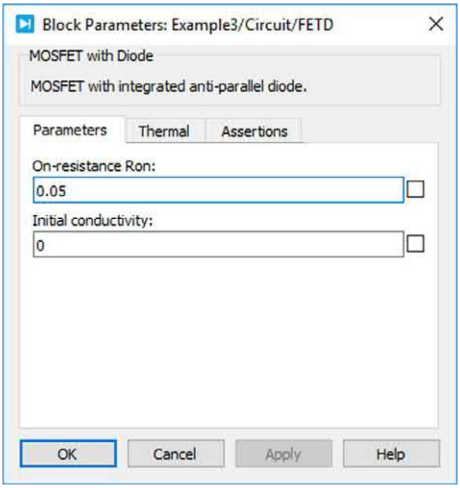

Example3/Circuit *
File Edit View Simulation Format Window Help
FETD
L
L: 220e-6
rL
R: .05
V_dc
V: 10
D1
rC
R: 0.05
C
C: 100e-6
RL
R: 8
Vm1
V
1
Vo
Scope
1
Duty
m
s
Sawtooth PWM

FIG. 4.92 Schematic of the buck converter. The 5-V output voltage is needed.

The settings for diode, MOSFET, and Sawtooth PWM generation block are shown in Figs. 4.93–4.95.


Block Parameters: Example3/Circuit/FETD
Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

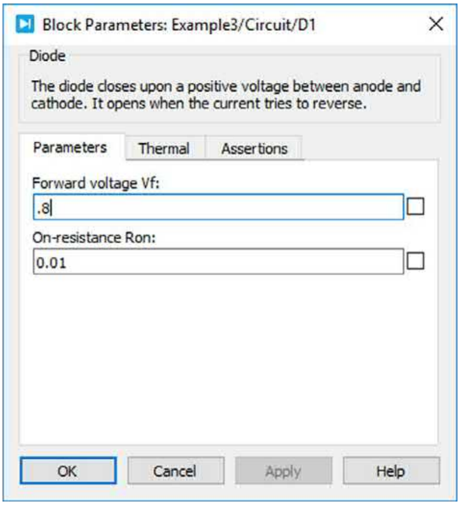
MOSFET with Diode

MOSFET with integrated anti-parallel diode.

On-resistance Ron: 0.05 ☐

Initial conductivity: 0 ☐

FIG. 4.93 Settings of MOSFET block.


Block Parameters: Example3/Circuit/D1

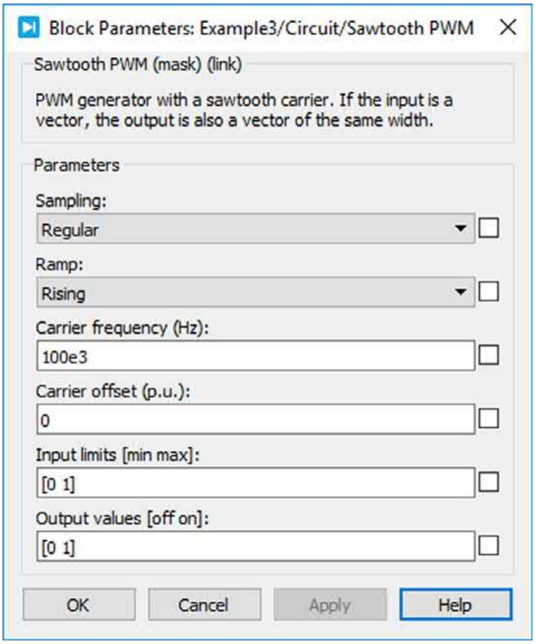
Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

Forward voltage Vf: .8 ☐

On-resistance Ron: 0.01 ☐

FIG. 4.94 Settings of diode block.


Block Parameters: Example3/Circuit/Sawtooth PWM

Sawtooth PWM (mask) (link)

PWM generator with a sawtooth carrier. If the input is a vector, the output is also a vector of the same width.

Parameters

Sampling: Regular ☐

Ramp: Rising ☐

Carrier frequency (Hz): 100e3 ☐

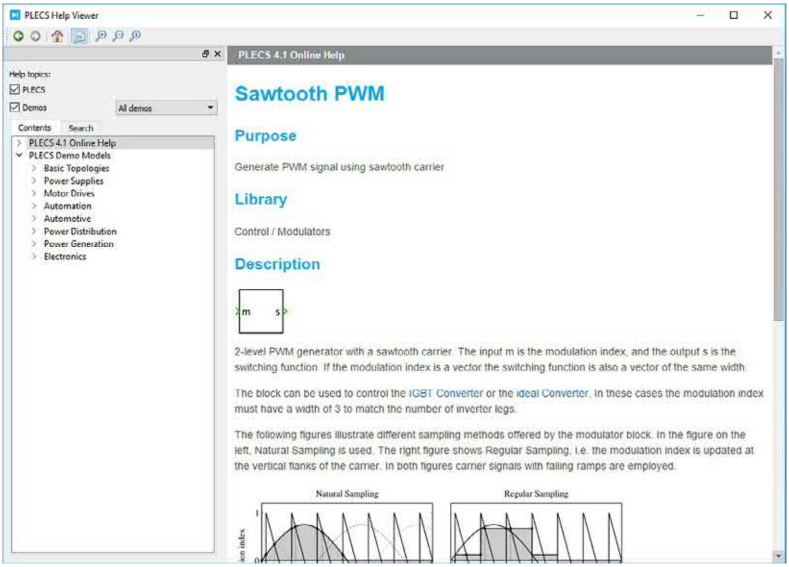
Carrier offset (p.u.): 0 ☐

Input limits [min max]: [0 1] ☐

Output values [off on]: [0 1] ☐

FIG. 4.95 Sawtooth PWM block settings.

When you need to see the details behind a block, you can double click on the block and click the Help button. For instance, if you click the Help button in Fig. 4.96, the window shown in Fig. 4.97 will appear. This window shows the detail of Sawtooth PWM generation block.


PLECS Help Viewer
Help topics:
☑ PLECS
☑ Demos
All demos
Contents
Search
PLECS 4.1 Online Help
PLECS Demo Models
Basic Topologies
Power Supplies
Motor Drives
Automation
Automotive
Power Distribution
Power Generation
Electronics
PLECS 4.1 Online Help
m
s
Natural Sampling
Regular Sampling
1
0
ion index

Sawtooth PWM

Purpose

Generate PWM signal using sawtooth carrier

Library

Control / Modulators

Description

2-level PWM generator with a sawtooth carrier. The input m is the modulation index, and the output s is the switching function. If the modulation index is a vector the switching function is also a vector of the same width.

The block can be used to control the IGBT Converter or the Ideal Converter. In these cases the modulation index must have a width of 3 to match the number of inverter legs.

The following figures illustrate different sampling methods offered by the modulator block. In the figure on the left, Natural Sampling is used. The right figure shows Regular Sampling, i.e. the modulation index is updated at the vertical flanks of the carrier. In both figures carrier signals with falling ramps are employed.

FIG. 4.96 Help of Sawtooth PWM block.

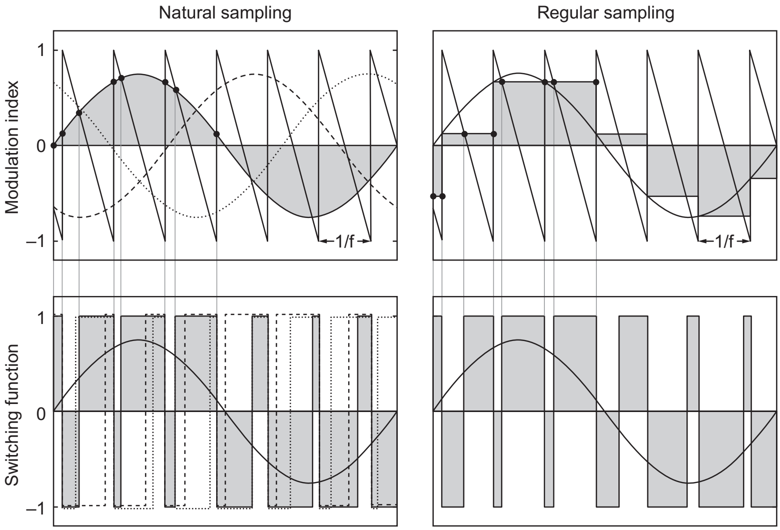


FIG. 4.97 Comparison between Natural Sampling and Regular Sampling. The modulating signal is shown with a sine wave. The carrier is the saw tooth signal.

Sampling (see Fig. 4.95) can be Natural or Regular. As shown in Fig. 4.97, in regular sampling, the modulating signal is sampled at the beginning of switching interval and doesn't change within one period. In natural sampling, the modulating signal is not sampled. The natural sampling is a good option if you plan to use an analog modulator. If you want to use a digital modulator, use the Regular Sampling.

Add the Impulse Response Analysis block to the Simulink file as shown in Fig. 4.98. The Impulse Response Analysis block can be found in the Analysis Tools section of the Simulink Library Browser (Fig. 4.99).

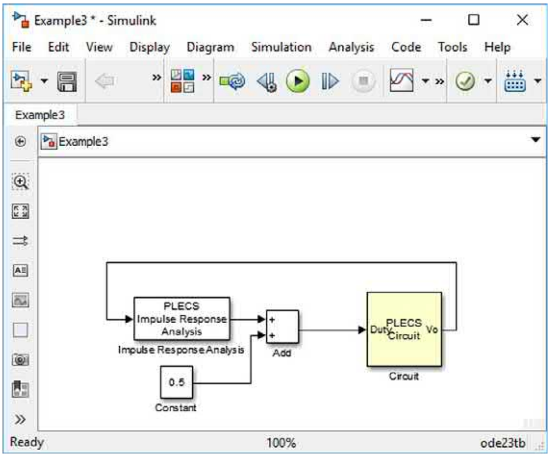

Example3 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example3
Example3
PLECS Impulse Response Analysis
Impulse Response Analysis
+
+
Add
0.5
Constant
PLECS Circuit
Duty
Vo
Circuit
Ready
100%
ode23tb

FIG. 4.98 Completed simulation to extract the small signal control-to-output transfer function. The output of 5 V is obtainable with duty ratio of 0.5.

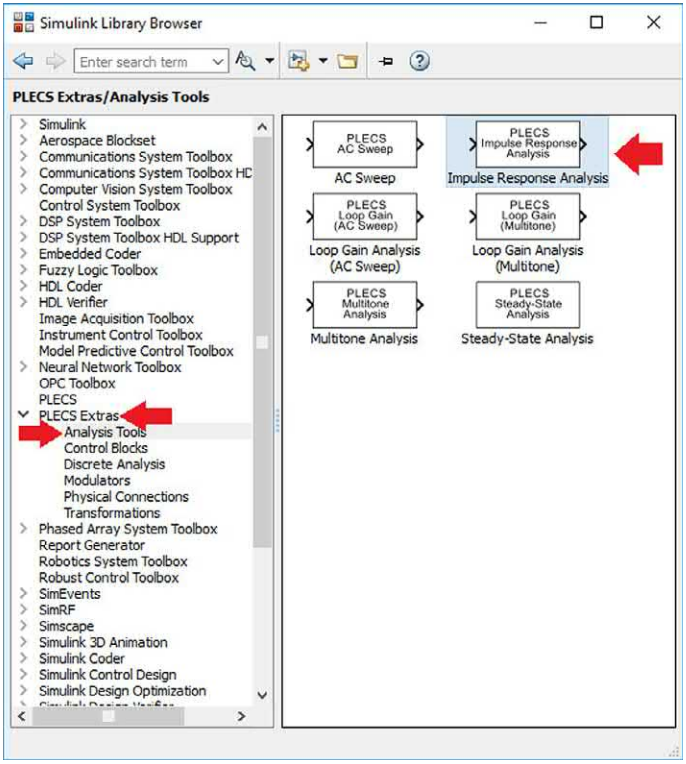
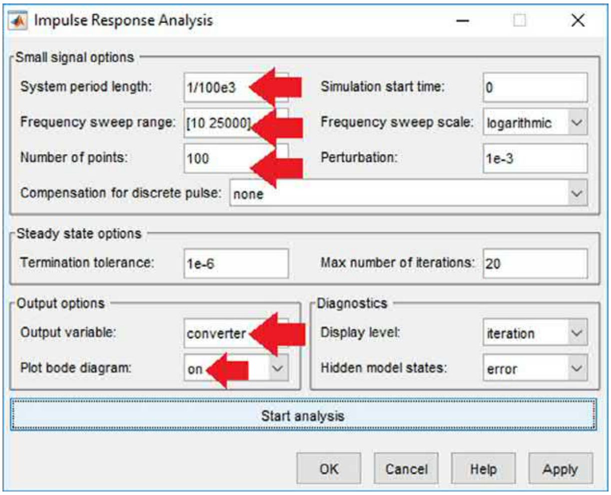

Simulink Library Browser
Enter search term
PLECS Extras/Analysis Tools
Simulink
Aerospace Blockset
Communications System Toolbox
Communications System Toolbox HD
Computer Vision System Toolbox
Control System Toolbox
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Analysis Tools
Control Blocks
Discrete Analysis
Modulators
Physical Connections
Transformations
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
PLECS AC Sweep
AC Sweep
PLECS Impulse Response Analysis
Impulse Response Analysis
PLECS Loop Gain (AC Sweep)
Loop Gain Analysis (AC Sweep)
PLECS Loop Gain (Multitone)
Loop Gain Analysis (Multitone)
PLECS Multitone Analysis
Multitone Analysis
PLECS Steady-State Analysis
Steady-State Analysis

FIG. 4.99 The Impulse Response Analysis block.

Double click the Impulse Response Analysis block placed in the Simulink environment. The window shown in Fig. 4.100 will appear.



Small signal options

System period length: 1/100e3

Frequency sweep range: [10 25000]

Number of points: 100

Compensation for discrete pulse: none

Simulation start time: 0

Frequency sweep scale: logarithmic

Perturbation: 1e-3

Steady state options

Termination tolerance: 1e-6

Max number of iterations: 20

Output options

Output variable: converter

Plot bode diagram: on

Diagnostics

Display level: iteration

Hidden model states: error

FIG. 4.100 Impulse Response Analysis block settings.

Fill the System period length box with 1/f, where f is the switching frequency of converter. Enter the desired frequency range, i.e., frequency range of output graph, in the Frequency sweep range box. The Impulse Response Analysis produces a logarithmically spaced frequency vector according to the values entered to Frequency sweep range and Number of points boxes. The Number of points box is the length of produced frequency vector. For instance according to the values shown in Fig. 4.100, the produced frequency vector will have 100 members, the minimum member is 10, and the maximum member is 25000. The frequency vector values can be obtained with this MATLAB command: `logspace(log(10),log10(25e3),100)`. Increasing the value entered into the Number of points box will produce a smoother graph; however, the analysis takes more time to be done.

After the user presses the Start analysis button, the result shown in Fig. 4.101 will appear. Note that the unit of frequency is Hertz in this plot.

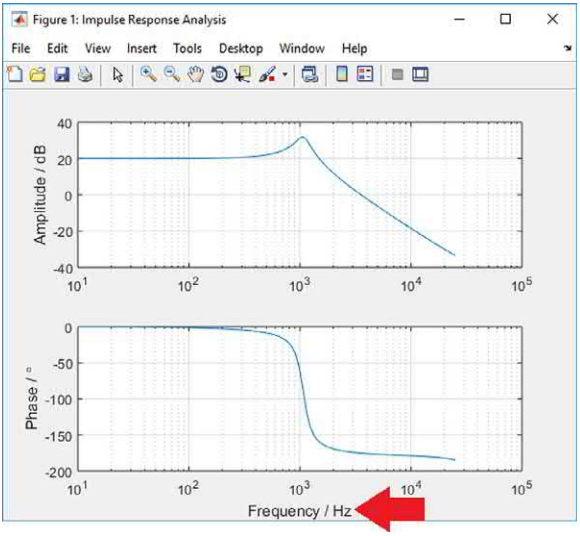


Figure 1: Impulse Response Analysis
File Edit View Insert Tools Desktop Window Help
Amplitude / dB
40
20
0
-20
-40
10^1
10^2
10^3
10^4
10^5
Phase / °
0
-50
-100
-150
-200
Frequency / Hz

FIG. 4.101 Result of analysis. The horizontal axis has the unit of Hertz.

Use the Data Cursor icon (Fig. 4.102) to read the values of different points of the graph. After you clicked the Data Cursor icon, click the point, which you want to read its value (Fig. 4.103). The value is shown in a box.

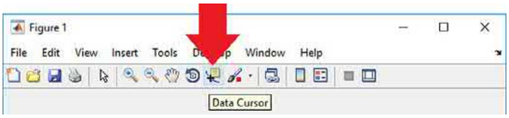


Figure 1
File Edit View Insert Tools Desktop Window Help
Data Cursor

FIG. 4.102 The Data Cursor icon.

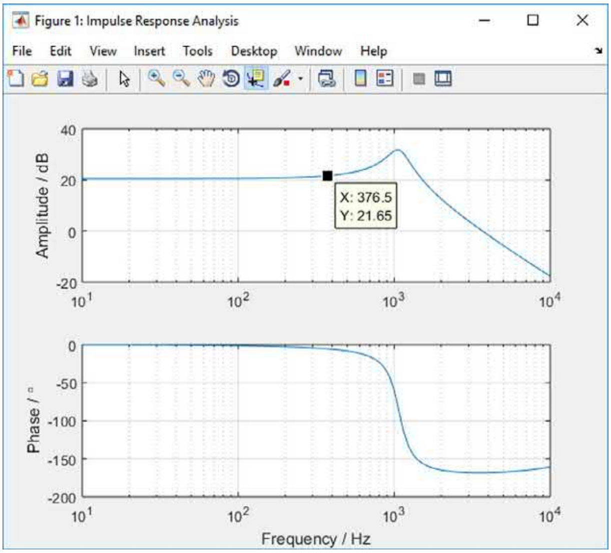


Figure 1: Impulse Response Analysis
File Edit View Insert Tools Desktop Window Help
Amplitude / dB
40
20
0
-20
10^1
10^2
10^3
10^4
X: 376.5
Y: 21.65
Phase / °
0
-50
-100
-150
-200
Frequency / Hz

FIG. 4.103 Addition of a datatip to the graph helps the user to read the values more easily.

You can move the datatip added to the graph. In order to make the movement smooth, right click on the data tip and select the Mouse Position (Fig. 4.104). If you want to remove the added datatip, right click on it and select the Delete Current Datatip.

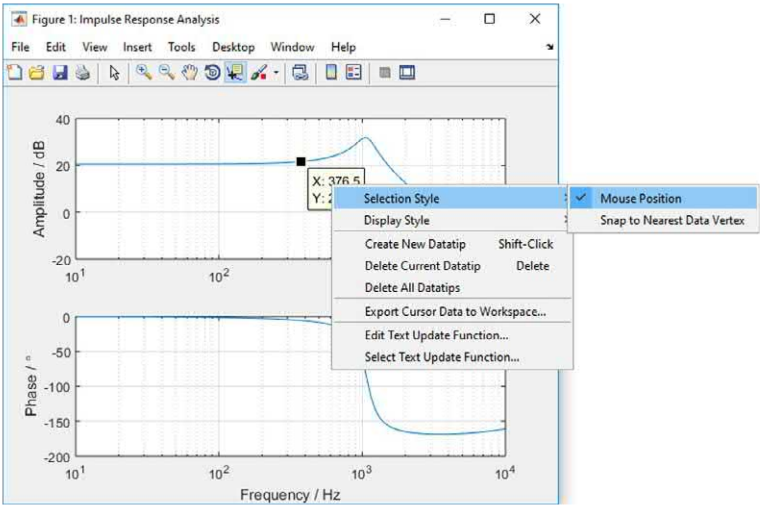


Figure 1: Impulse Response Analysis
File Edit View Insert Tools Desktop Window Help
Amplitude / dB
40
20
0
-20
X: 376.5
Y:
Selection Style
Display Style
Create New Datatip Shift-Click
Delete Current Datatip Delete
Delete All Datatips
Export Cursor Data to Workspace...
Edit Text Update Function...
Select Text Update Function...
✓ Mouse Position
Snap to Nearest Data Vertex
Phase / °
0
-50
-100
-150
-200
10^1
10^2
10^3
10^4
Frequency / Hz

FIG. 4.104 Select Mouse Position to move the added data tip more smoothly.

You can save the obtained graph as graphic file. In order to do this, click the Save As … (Fig. 4.105)

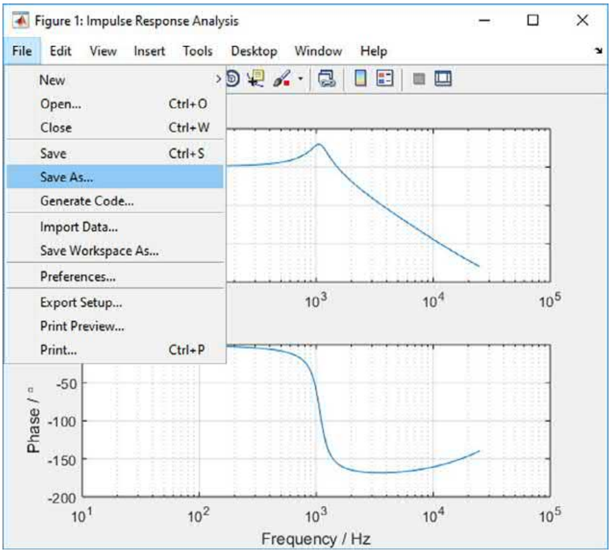
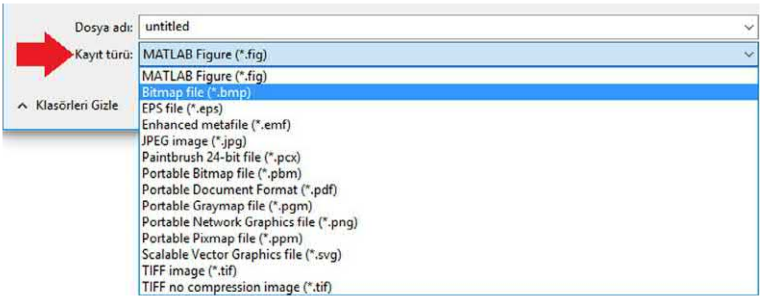


Figure 1: Impulse Response Analysis
File Edit View Insert Tools Desktop Window Help
New
Open... Ctrl+O
Close Ctrl+W
Save Ctrl+S
Save As...
Generate Code...
Import Data...
Save Workspace As...
Preferences...
Export Setup...
Print Preview...
Print... Ctrl+P
10^3
10^4
10^5
Phase / °
-50
-100
-150
-200
10^1
10^2
10^3
10^4
10^5
Frequency / Hz

FIG. 4.105 Use Save As… to export the obtained graph as a graphical file.

Set the desired path, name, and output file format in the appeared window (Fig. 4.106).



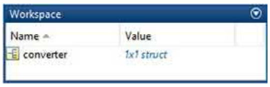
Dosya adı: untitled

Kayıt türü: MATLAB Figure (*.fig)

Klasörleri Gizle
MATLAB Figure (*.fig)
Bitmap file (*.bmp)
EPS file (*.eps)
Enhanced metafile (*.emf)
JPEG image (*.jpg)
Paintbrush 24-bit file (*.pcx)
Portable Bitmap file (*.pbm)
Portable Document Format (*.pdf)
Portable Graymap file (*.pgm)
Portable Network Graphics file (*.png)
Portable Pixmap file (*.ppm)
Scalable Vector Graphics file (*.svg)
TIFF image (*.tif)
TIFF no compression image (*.tif)

FIG. 4.106 Selection of output file format.

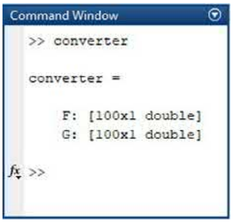
If you notice to the Workspace, you will see that a new variable is added to it (Fig. 4.107). The variable name is the same as the one written in Output variable box (see Fig. 4.100).



Name	Value
converter	1x1 struct

FIG. 4.107 After analysis is done, a new variable is added to the Workspace.

The variable `converter` has two fields: F and G. The field F keeps the frequency and G is the response of converter. F is real vector and G is a complex vector (Fig. 4.108).

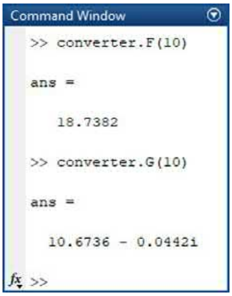


Command Window

```
>> converter

converter =

    F: [100x1 double]
    G: [100x1 double]

>>
```

FIG. 4.108 Fields of the variable `converter`.

For instance, according to Fig. 4.109, the value of 10th element of F and G is 18.7382 and 10.6736 – 0.0442i, respectively.



```
>> converter.F(10)

ans =

   18.7382

>> converter.G(10)

ans =

  10.6736 - 0.0442i

>>
```

FIG. 4.109 Values of 10th elements of F and G. Note that $10.6736-0.0442i=10.6737e^{-j\times 0.2372^\circ}$.

This means at 18.7382 Hz ($=18.7382\times 2\pi=117.7356\,\frac{\text{Rad}}{\text{s}}$), value of magnitude plot is $20\times\log(10.6737)=20.5663$ db and value of phase plot is $-0.2372°$. You can add cursor to the obtained graph and check this calculation (Fig. 4.110).

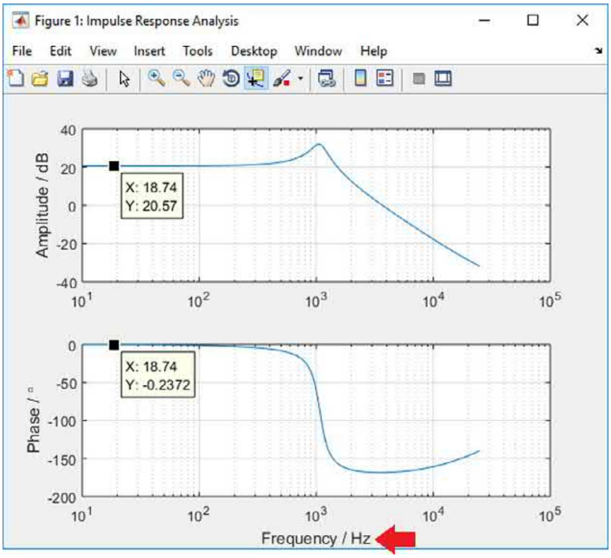
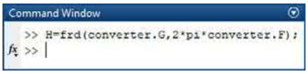


FIG. 4.110 Magnitude and phase values at 18.74 Hz.

4.8 Fitting a transfer function to obtained graph

The result of previous analysis was graphical, i.e., the software didn't give us the equation of transfer function. You can use the System Identification Toolbox® to fit a transfer function to the obtained data.

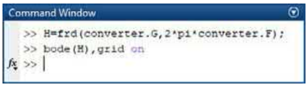
In order to estimate a transfer function for the obtained data, the frequency response data model (frd object) of obtained data are formed (Fig. 4.111). Since the `converter.F` is in Hertz, it must be multiplied with 2π in order to obtain it in $\frac{\text{Rad}}{\text{s}}$.



```
>> H=frd(converter.G,2*pi*converter.F);
>> |
```

FIG. 4.111 Formation of frd object.

You can draw the Bode diagram of frd model with the aid of `bode` command (Fig. 4.112). The result is shown in Fig. 4.113.



```
>> H=frd(converter.G,2*pi*converter.F);
>> bode(H),grid on
>>
```

FIG. 4.112 Drawing the frequency response of frd object H.

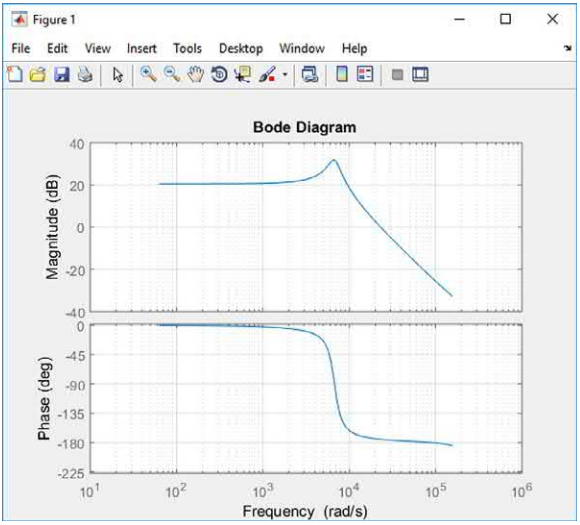
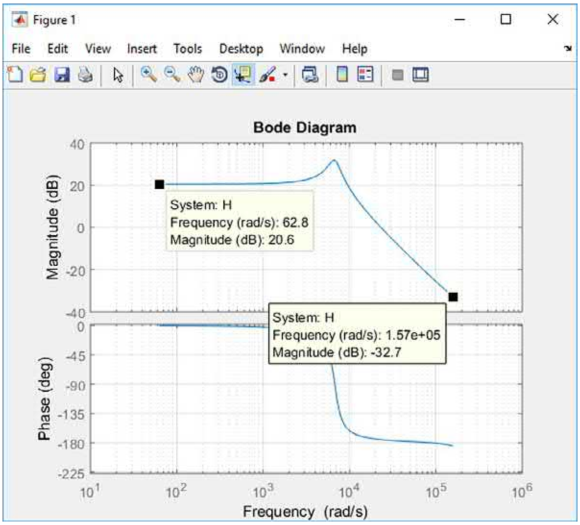


FIG. 4.113 Frequency response of H. The vertical axis of graph has the unit of $\frac{\text{Rad}}{\text{s}}$.

As shown in Fig. 4.114, the graph starts from $62.8\,\frac{\text{Rad}}{\text{s}}$ to $1.57 \times 10^5\,\frac{\text{Rad}}{\text{s}}$ (10 Hz to 25 kHz).

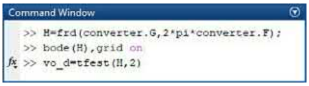


System: H
Frequency (rad/s): 62.8
Magnitude (dB): 20.6

System: H
Frequency (rad/s): 1.57e+05
Magnitude (dB): -32.7

FIG. 4.114 The frequency response graph of H starts from $62.8\,\frac{\text{Rad}}{\text{s}} = 10\,\text{Hz}$ and ends at $1.57 \times 10^5\,\frac{\text{Rad}}{\text{s}} = 25\,\text{kHz}$.

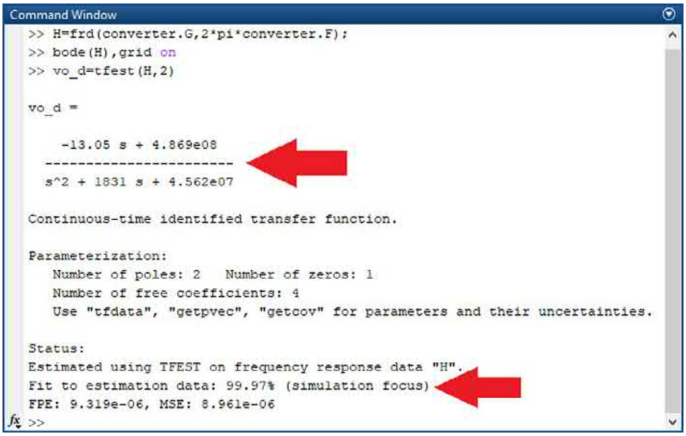
You can estimate a transfer function for the frd object H with the aid of `tfest` command. We want to find a second-order estimation since the buck converter contains two independent energy storage elements (a capacitor and an inductor). Result is shown in Fig. 4.115.



```
  >> H=frd(converter.G,2*pi*converter.F);
  >> bode(H),grid on
fx >> vo_d=tfest(H,2)
```

FIG. 4.115 Estimation of a second-order controller to frd object H.

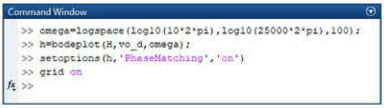
The Fit to estimation data (Fig. 4.116) shows the goodness of fit. Since it is near to 100%, the estimation overlaps the original frequency response very good.



```
>> H=frd(converter.G,2*pi*converter.F);
>> bode(H),grid on
>> vo_d=tfest(H,2)

vo_d =

    -13.05 s + 4.869e08
  -----------------------
  s^2 + 1831 s + 4.562e07

Continuous-time identified transfer function.

Parameterization:
   Number of poles: 2   Number of zeros: 1
   Number of free coefficients: 4
   Use "tfdata", "getpvec", "getcov" for parameters and their uncertainties.

Status:
Estimated using TFEST on frequency response data "H".
Fit to estimation data: 99.97% (simulation focus)
FPE: 9.319e-06, MSE: 8.961e-06
fx >>
```

FIG. 4.116 Estimated transfer function.

The code shown in Fig. 4.117 compares the frequency response of original system (i.e., the frequency response produced by PLECS) with the estimated one. Result is shown in Fig. 4.118.



```
  >> omega=logspace(log10(10*2*pi),log10(25000*2*pi),100);
  >> h=bodeplot(H,vo_d,omega);
  >> setoptions(h,'PhaseMatching','on')
  >> grid on
fx >>
```

FIG. 4.117 Comparison of the frequency response of estimated transfer function and the original system.

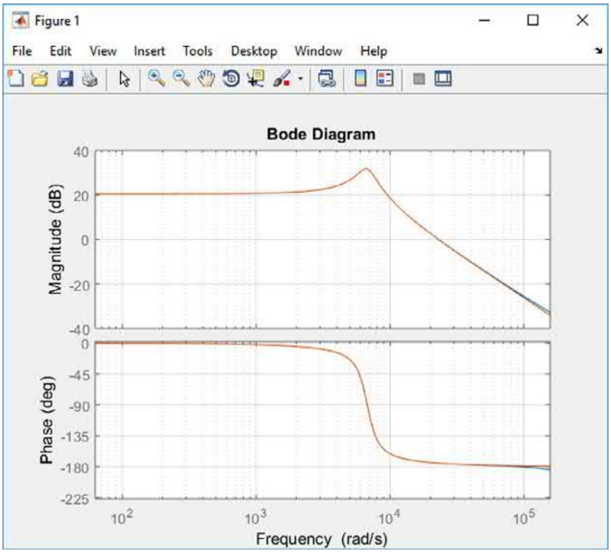
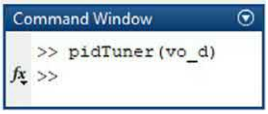


FIG. 4.118 Comparison results. The two curves overlap almost everywhere.

4.9 Designing a controller

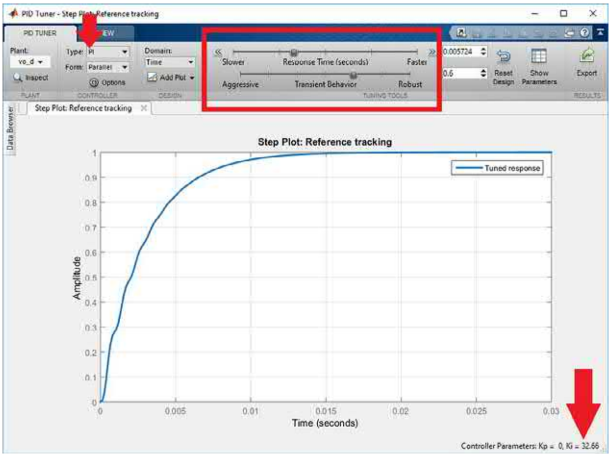
We design a PI controller for the estimated transfer function. MATLAB® has a powerful command (`pidTuner`) to design PID controllers. Enter the command shown in Fig. 4.119 to start the design process.



```
>> pidTuner(vo_d)
>>
```

FIG. 4.119 Running the pidTuner application of MATLAB.

After you entered the `pidTuner(vo_d)`command, the window shown in Fig. 4.120 will appear. Use the Type to select the desired controller type (i.e., P, I, PI, I, PID). Move the sliders until you obtain the desired response. The controller parameters are shown in the right button section of window.


PID Tuner - Step Plot: Reference tracking
PID TUNER
VIEW
Plant:
vo_d
Inspect
Type: PI
Form: Parallel
Options
Domain:
Time
Add Plot
Slower
Response Time (seconds)
Faster
Aggressive
Transient Behavior
Robust
0.005724
0.6
Reset Design
Show Parameters
Export
PLANT
CONTROLLER
DESIGN
TUNING TOOLS
RESULTS
Data Browser
Step Plot: Reference tracking
Step Plot: Reference tracking
Tuned response
Amplitude
Time (seconds)
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1
0.005
0.01
0.015
0.02
0.025
0.03

Controller Parameters: Kp = 0, Ki = 32.66

FIG. 4.120 The PID Tuner window. Change the sliders until you obtain the desired result.

You can tune the controller with the frequency domain criteria (i.e., phase margin and gain margin) as well. In order to do this, select the Frequency (Fig. 4.121).

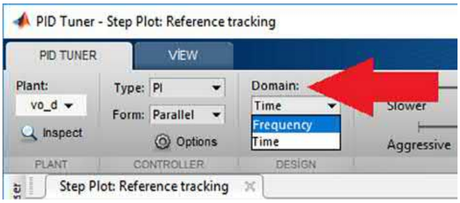

PID Tuner - Step Plot: Reference tracking
PID TUNER
VIEW
Plant:
vo_d
Inspect
Type: PI
Form: Parallel
Options
Domain:
Time
Frequency
Time
Slower
Aggressive
PLANT
CONTROLLER
DESIGN
Step Plot: Reference tracking

FIG. 4.121 The tuning can be done in the frequency domain.

The response shown in Fig. 4.120 seems good. It is not oscillatory and the settling time is 15 ms. We want to simulate the buck converter with this controller. In order to simulate the closed-loop buck converter, the simulation diagram shown is changed like Fig. 4.122.

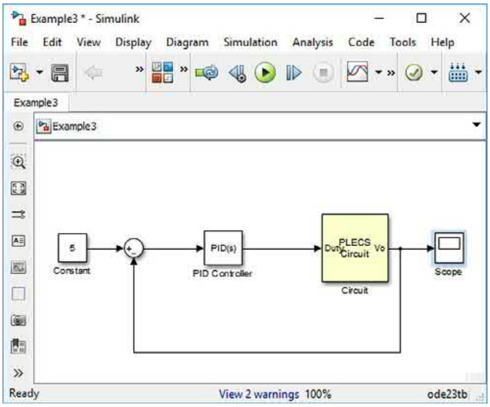
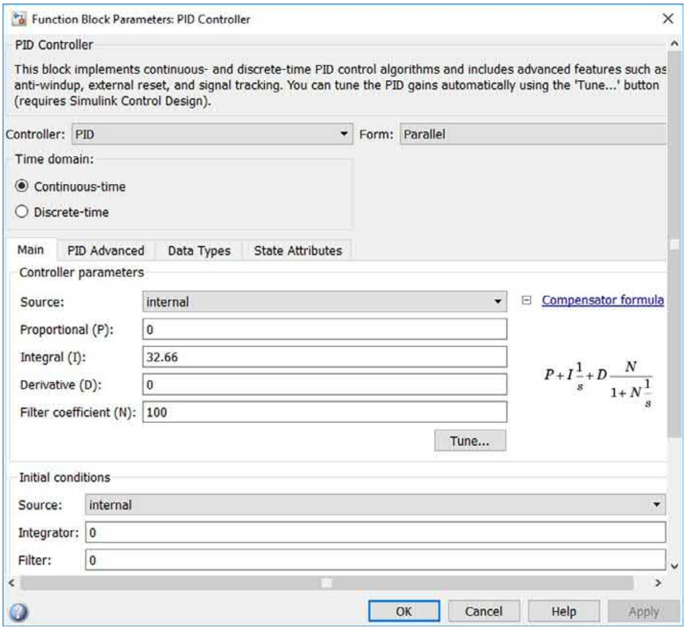

Example3 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example3
Example3
5
Constant
PID(s)
PID Controller
Duty
PLECS Circuit
Vo
Circuit
Scope
Ready
View 2 warnings
100%
ode23tb

FIG. 4.122 The completed Simulink model. Kp = 0, Ki = 32.66, Kd = 0 for PID Controller block.

Setting of PID controller block is shown in Fig. 4.123. The simulation result is shown in Fig. 4.124. As shown, the system tracks the command with zero steady-state error.



Function Block Parameters: PID Controller

PID Controller

This block implements continuous- and discrete-time PID control algorithms and includes advanced features such as anti-windup, external reset, and signal tracking. You can tune the PID gains automatically using the 'Tune...' button (requires Simulink Control Design).

Controller: PID   Form: Parallel

Time domain:

● Continuous-time

○ Discrete-time

Controller parameters

Source: internal

Proportional (P): 0

Integral (I): 32.66

Derivative (D): 0

Filter coefficient (N): 100

Compensator formula

$$P+I\frac{1}{s}+D\frac{N}{1+N\frac{1}{s}}$$

Initial conditions

Source: internal

Integrator: 0

Filter: 0

FIG. 4.123 PID controller block settings.

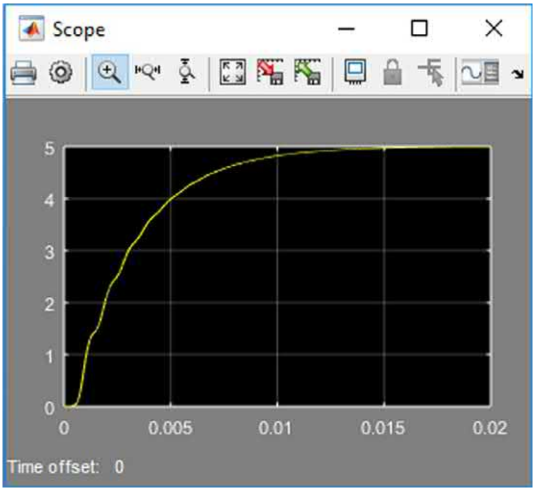


FIG. 4.124 Simulation result.

You can study the effect of change in reference command with the simulation diagram shown in Fig. 4.125. The value of reference command in this simulation changes from 5 V to 8 V at 20 ms. The step block settings is shown in Fig. 4.126. The simulation result is shown in Fig. 4.127.

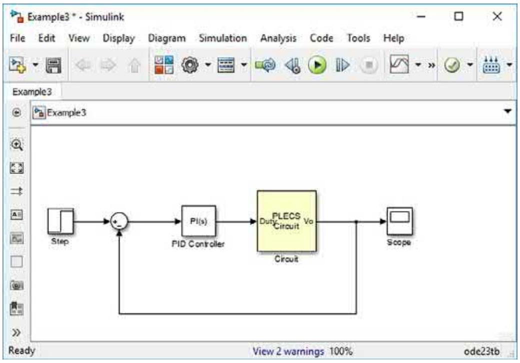
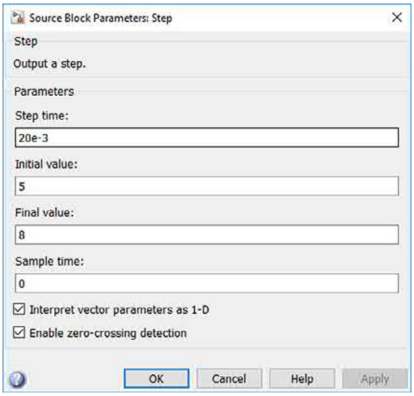

Example3 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example3
Example3
Step
PI(s)
PID Controller
Duty PLECS Circuit Vo
Circuit
Scope
Ready
View 2 warnings 100%
ode23tb

FIG. 4.125 The reference of feedback loop changes from 5 V to 8 V at $t = 20$ ms.



Source Block Parameters: Step

Step

Output a step.

Parameters

Step time:
20e-3

Initial value:
5

Final value:
8

Sample time:
0

☑ Interpret vector parameters as 1-D

☑ Enable zero-crossing detection

OK
Cancel
Help
Apply

FIG. 4.126 Setting of Step block.

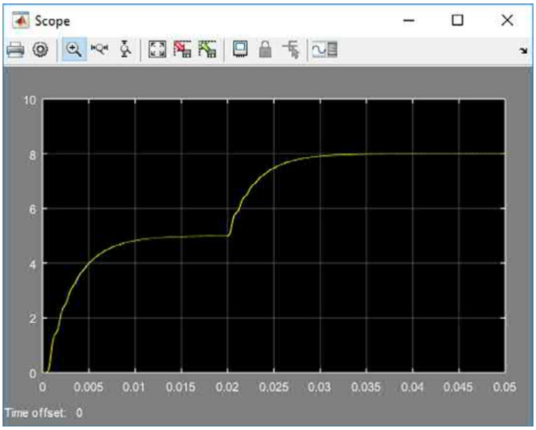


FIG. 4.127 Simulation result.

You can study the effect of change in output load as well. In order to do this, the schematic is changed as shown in Fig. 4.128. Before closing the FETD1, the output load is 8 Ω. When the step block forces the FETD1 to conduct, the output load changes to $\frac{8\times 8}{8+8}=4\,\Omega$. The step block settings are shown in Fig. 4.129. Simulation result is shown in Fig. 4.130. As shown, the controller keeps the voltage constant despite of disturbance (i.e., change in output load).

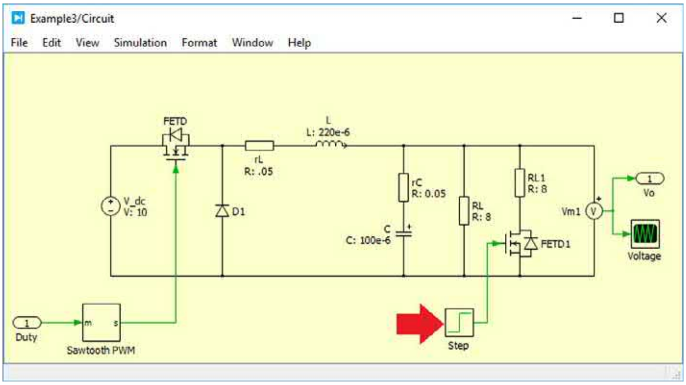
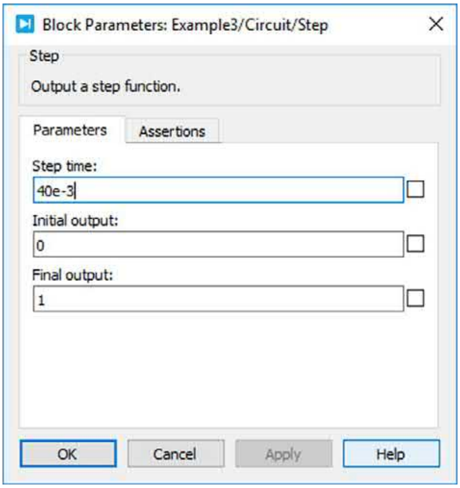

Example3/Circuit
File Edit View Simulation Format Window Help
FETD
L
L: 220e-6
rL
R: .05
V_dc
V: 10
D1
rC
R: 0.05
C
C: 100e-6
RL
R: 8
RL1
R: 8
Vm1
V
FETD1
1
Vo
Voltage
1
Duty
m
s
Sawtooth PWM
Step

FIG. 4.128 Schematic to study the effect of change in output load.


Block Parameters: Example3/Circuit/Step
Parameters
Assertions
OK
Cancel
Apply
Help

Step

Output a step function.

Step time: 40e-3 ☐

Initial output: 0 ☐

Final output: 1 ☐

FIG. 4.129 The Step block settings.

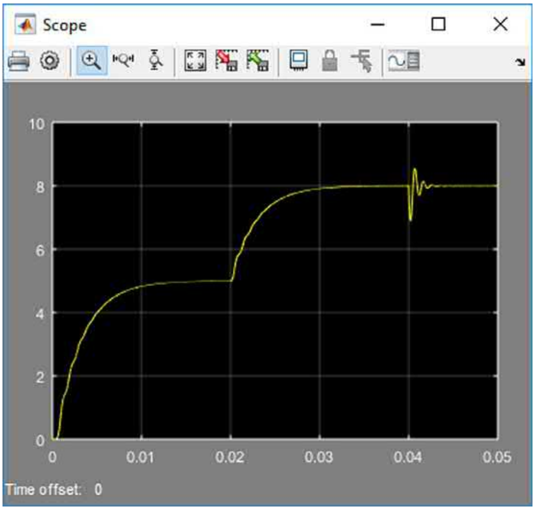


FIG. 4.130 The simulation result.

It is recommended to use the scope block provided by PLECS block set to see the voltage/currents (Fig. 4.131). The PLECS scope provides much more capability to see voltage/currents in comparison to the general-purpose scope block of Simulink®. The PLECS scope can be found in the PLECS section of Simulink library browser window (Fig. 4.132).

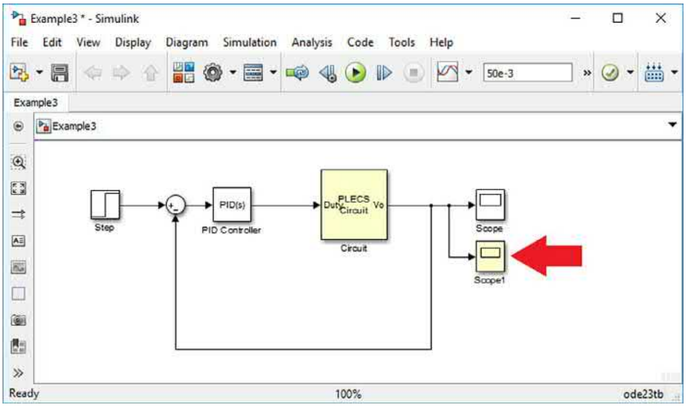

Example3 * - Simulink
File
Edit
View
Display
Diagram
Simulation
Analysis
Code
Tools
Help
50e-3
Example3
Example3
Step
PID(s)
PID Controller
PLECS Circuit
Duty
Vo
Circuit
Scope
Scope1
Ready
100%
ode23tb

FIG. 4.131 PLECS Scope block is a better option to analyze the voltage/currents.

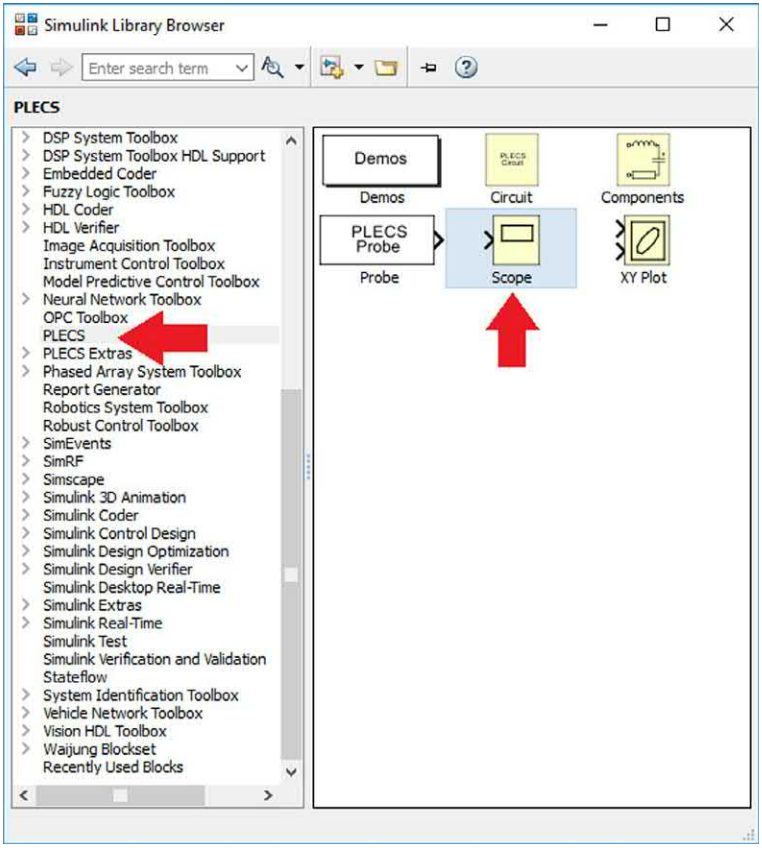

Simulink Library Browser
Enter search term
PLECS
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Desktop Real-Time
Simulink Extras
Simulink Real-Time
Simulink Test
Simulink Verification and Validation
Stateflow
System Identification Toolbox
Vehicle Network Toolbox
Vision HDL Toolbox
Waijung Blockset
Recently Used Blocks
Demos
Demos
Circuit
Components
PLECS Probe
Probe
Scope
XY Plot

FIG. 4.132 The Scope block of PLECS blockset.

You can test the designed controller with another type of disturbance as well. The schematic shown in Fig. 4.133 changes the input voltage from 10 V to 12 V at 45 ms. The step block settings are shown in Fig. 4.134.

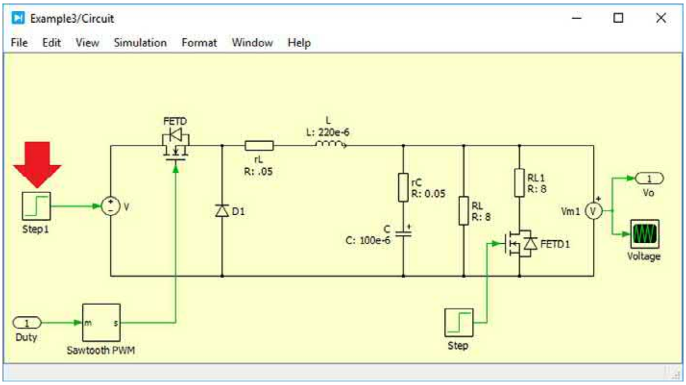
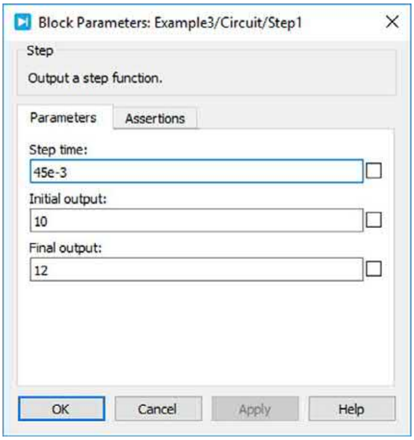

Example3/Circuit
File Edit View Simulation Format Window Help
FETD
L
L: 220e-6
rL
R: .05
D1
V
Step1
rC
R: 0.05
C
C: 100e-6
RL
R: 8
RL1
R: 8
FETD1
Vm1
V
1
Vo
Voltage
1
Duty
m
s
Sawtooth PWM
Step

FIG. 4.133 Schematic to study the effect of change in input voltage.



Block Parameters: Example3/Circuit/Step1

Step

Output a step function.

Step time: 45e-3 ☐

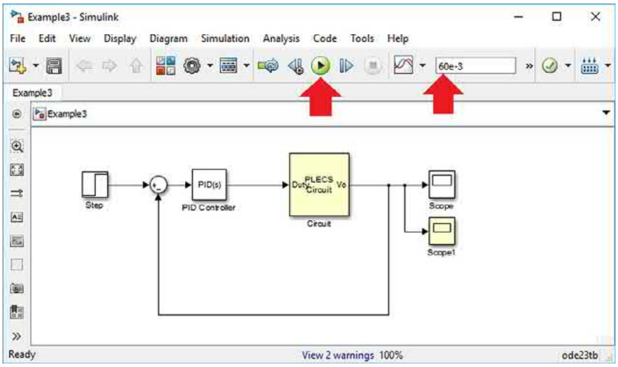
Initial output: 10 ☐

Final output: 12 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 4.134 Setting of the Step1 block.

Increase the simulation interval from 50 ms to 60 ms and run the simulation. The simulation result is shown in Fig. 4.135. As shown, the controller keeps the output voltage constant despite the change in the input voltage (Fig. 4.136).



Example3 - Simulink
File
Edit
View
Display
Diagram
Simulation
Analysis
Code
Tools
Help
60e-3
Example3
Example3
Step
PID(s)
PID Controller
Duty
PLECS Circuit
Vo
Circuit
Scope
Scope1
Ready
View 2 warnings
100%
ode23tb

FIG. 4.135 Increasing the simulation time to 60 ms. Click the Run button to run the simulation.

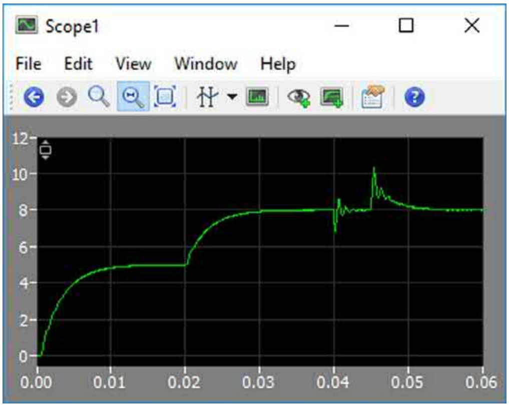


FIG. 4.136 Simulation result.

4.10 Obtaining the control-to-inductor current transfer function

You can obtain the transfer function between the control input (duty ratio) and inductor current with the schematic diagram shown in Fig. 4.137. The Simulink® model must be like Fig. 4.138.

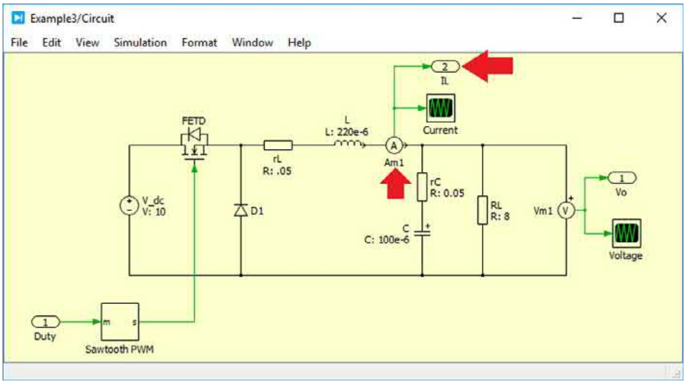

Example3/Circuit
File Edit View Simulation Format Window Help
2
IL
Current
FETD
L
L: 220e-6
A
Am1
rL
R: .05
V_dc
V: 10
D1
rC
R: 0.05
C
C: 100e-6
RL
R: 8
Vm1
V
1
Vo
Voltage
1
Duty
m
s
Sawtooth PWM

FIG. 4.137 Addition of an ampere meter and Signal Outport blocks to the schematic.

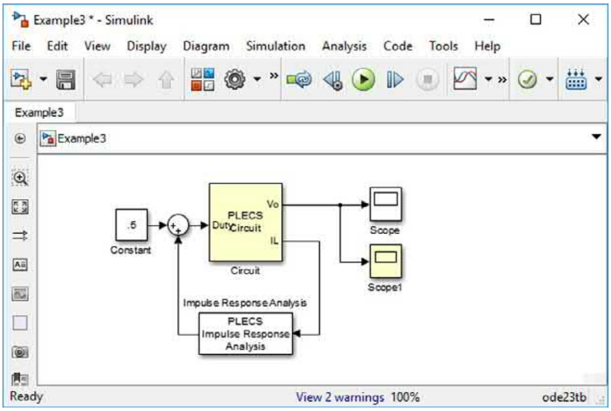
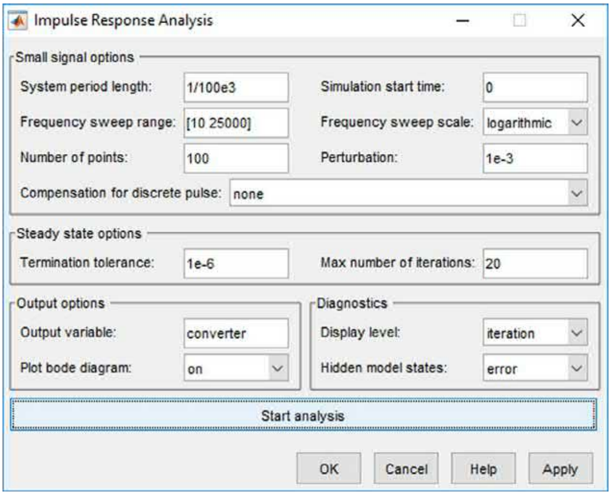

Example3 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example3
Example3
.5
Constant
PLECS
Circuit
Duty
Vo
IL
Circuit
Scope
Scope1
Impulse Response Analysis
PLECS
Impulse Response
Analysis
Ready
View 2 warnings 100%
ode23tb

FIG. 4.138 The Simulink model to extract the $\frac{\tilde{i}_L(s)}{\tilde{d}(s)}$.

Setting of PLECS Impulse Response Analysis block is shown in Fig. 4.139. We want to obtain the frequency response in the 10 Hz–25 kHz interval. Click the Start analysis button to start the analysis. The simulation result is shown in Fig. 4.140.



Small signal options

System period length: 1/100e3

Simulation start time: 0

Frequency sweep range: [10 25000]

Frequency sweep scale: logarithmic

Number of points: 100

Perturbation: 1e-3

Compensation for discrete pulse: none

Steady state options

Termination tolerance: 1e-6

Max number of iterations: 20

Output options

Output variable: converter

Plot bode diagram: on

Diagnostics

Display level: iteration

Hidden model states: error

FIG. 4.139 The Impulse Response Analysis block settings.

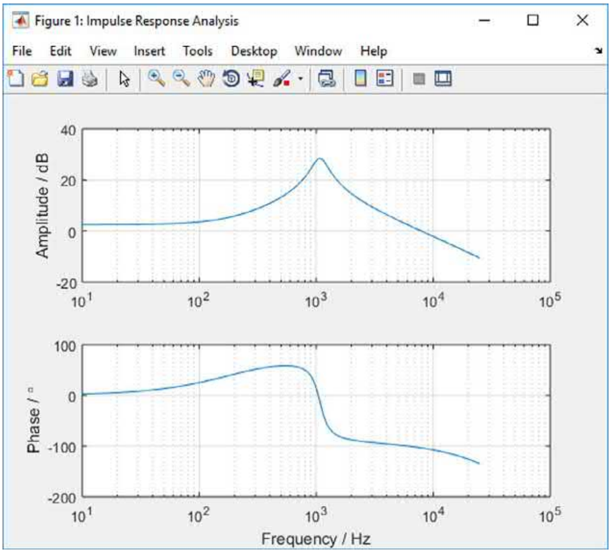


FIG. 4.140 Control-to-inductor current transfer function.

4.11 Extraction of output impedance

You can obtain the output impedance of converter with the aid of schematic shown in Fig. 4.141 and Simulink model shown in Fig 4.142.

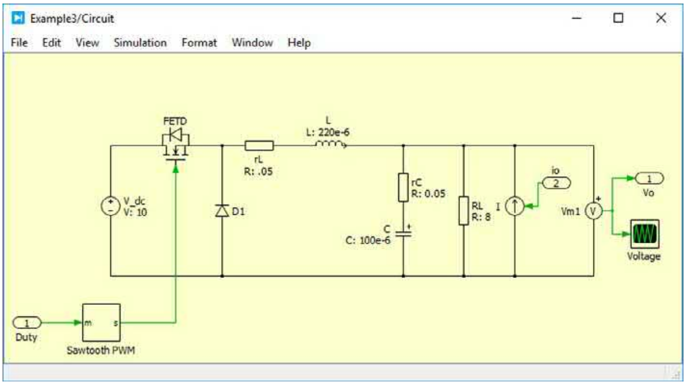

Example3/Circuit
File Edit View Simulation Format Window Help
FETD
L
L: 220e-6
rL
R: .05
V_dc
V: 10
D1
rC
R: 0.05
C
C: 100e-6
RL
R: 8
I
io
2
Vm1
V
1
Vo
Voltage
1
Duty
m
s
Sawtooth PWM

FIG. 4.141 The schematic to extract the output impedance.

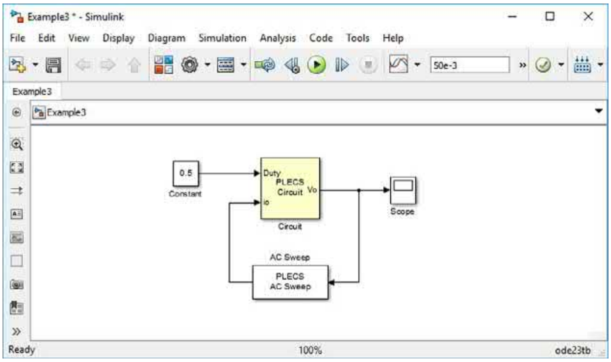

Example3 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
50e-3
Example3
Example3
0.5
Constant
Duty
PLECS
Circuit
Vo
io
Circuit
Scope
AC Sweep
PLECS
AC Sweep
Ready
100%
ode23tb

FIG. 4.142 The Simulink model to extract the output impedance.

The AC sweep block can be found in the PLECS Extras section of Simulink Library Browser (Fig. 4.143).

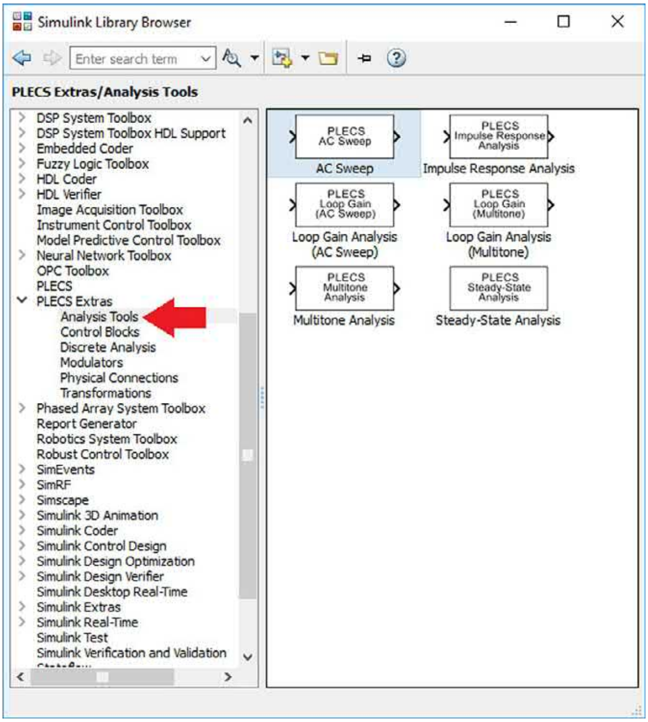
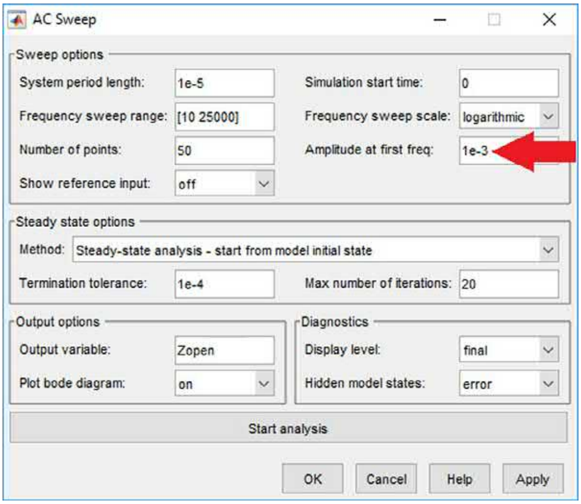

Simulink Library Browser
Enter search term
PLECS Extras/Analysis Tools
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Analysis Tools
Control Blocks
Discrete Analysis
Modulators
Physical Connections
Transformations
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Desktop Real-Time
Simulink Extras
Simulink Real-Time
Simulink Test
Simulink Verification and Validation
PLECS AC Sweep
AC Sweep
PLECS Impulse Response Analysis
Impulse Response Analysis
PLECS Loop Gain (AC Sweep)
Loop Gain Analysis (AC Sweep)
PLECS Loop Gain (Multitone)
Loop Gain Analysis (Multitone)
PLECS Multitone Analysis
Multitone Analysis
PLECS Steady-State Analysis
Steady-State Analysis

FIG. 4.143 AC sweep can be found in the Analysis Tools section of Simulink Library Browser.

Double click the AC sweep block and change the block settings like Fig. 4.144.


AC Sweep

Sweep options

System period length: 1e-5

Simulation start time: 0

Frequency sweep range: [10 25000]

Frequency sweep scale: logarithmic

Number of points: 50

Amplitude at first freq: 1e-3

Show reference input: off

Steady state options

Method: Steady-state analysis - start from model initial state

Termination tolerance: 1e-4

Max number of iterations: 20

Output options

Output variable: Zopen

Plot bode diagram: on

Diagnostics

Display level: final

Hidden model states: error

Start analysis
OK
Cancel
Help
Apply

FIG. 4.144 Setting of AC sweep block. It is recommended to click the Help button of AC sweep block and read the provided document.

Run the analysis by clicking the Start analysis. The result shown in Fig. 4.145 will appear.

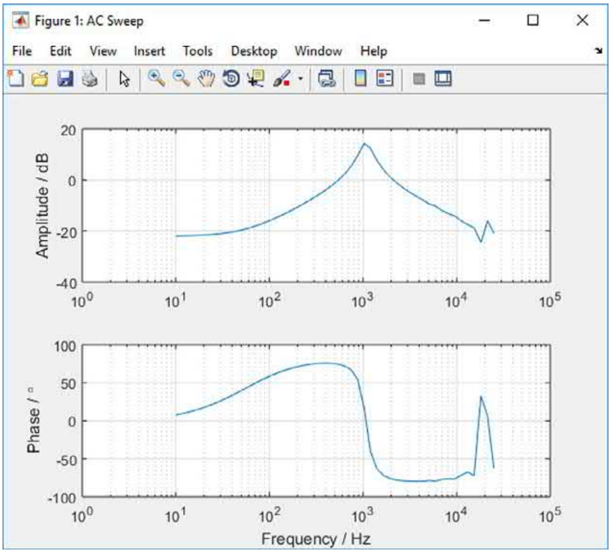


Figure 1: AC Sweep
File Edit View Insert Tools Desktop Window Help
Amplitude / dB
20
0
-20
-40
10^0 10^1 10^2 10^3 10^4 10^5
Phase / °
100
50
0
-50
-100
10^0 10^1 10^2 10^3 10^4 10^5
Frequency / Hz

FIG. 4.145 Simulation result for Amplitude at first freq.= 1 m.

The high-frequency portion of graph is not smooth, i.e., there are some fluctuations in the high-frequency portion of the graph. Increase the Amplitude at first freq. (Fig. 4.144) to 5e-3 and rerun the analysis. Fig. 4.146 shows the analysis result.

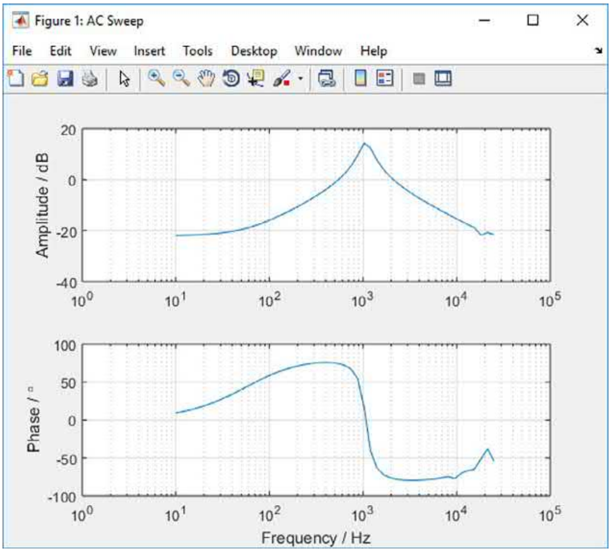


Figure 1: AC Sweep
File Edit View Insert Tools Desktop Window Help
Amplitude / dB
20
0
-20
-40
10^0 10^1 10^2 10^3 10^4 10^5
Phase / °
100
50
0
-50
-100
10^0 10^1 10^2 10^3 10^4 10^5
Frequency / Hz

FIG. 4.146 Simulation result for Amplitude at first freq.= 5 m.

As shown in Fig. 4.146, the curves are smoother in comparison to Fig. 4.145. If the Amplitude at first freq. (Fig. 4.144) increases to 10e-3, the result shown in Fig. 4.147 will appear. When there are fluctuations in the high-frequency portion of obtained transfer function, simply increase the Amplitude at first freq. in order to obtain a smoother graph.

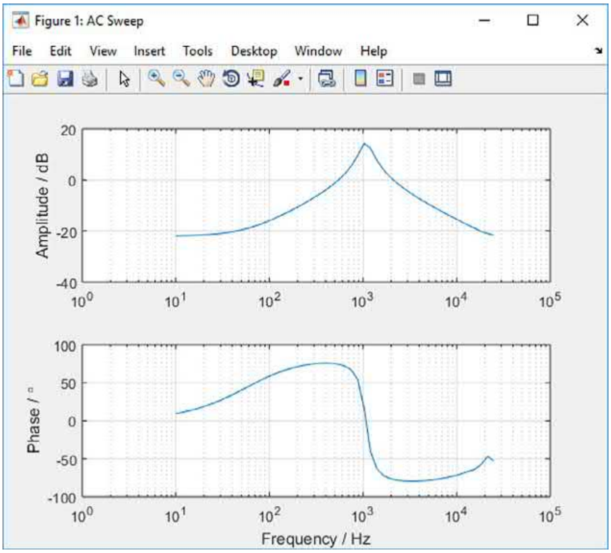


FIG. 4.147 Simulation result for Amplitude at first freq.= 10 m.

4.12 Steady-state analysis

Consider a schematic diagram and Simulink model like the one shown in Figs. 4.148 and 4.149, respectively. We want to obtain the steady-state values of inductor current and output voltage.

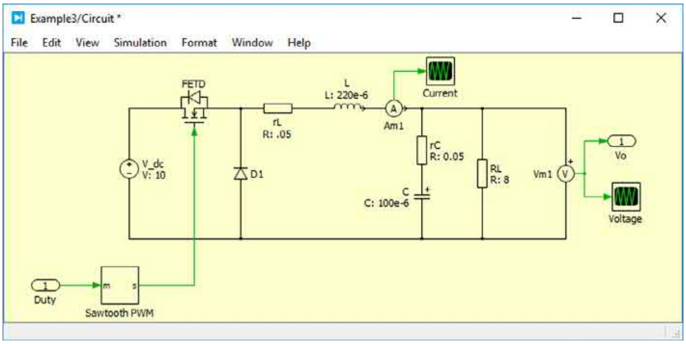

Example3/Circuit *
File Edit View Simulation Format Window Help
FETD
L
L: 220e-6
Current
rL
R: .05
Am1
A
rC
R: 0.05
V_dc
V: 10
D1
RL
R: 8
Vm1
V
1
Vo
C
C: 100e-6
Voltage
1
Duty
m
s
Sawtooth PWM

FIG. 4.148 Schematic of buck converter. The output voltage of 5 V is needed.

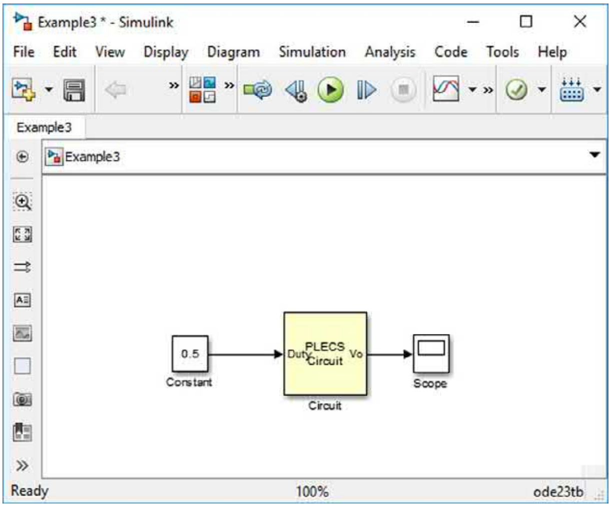
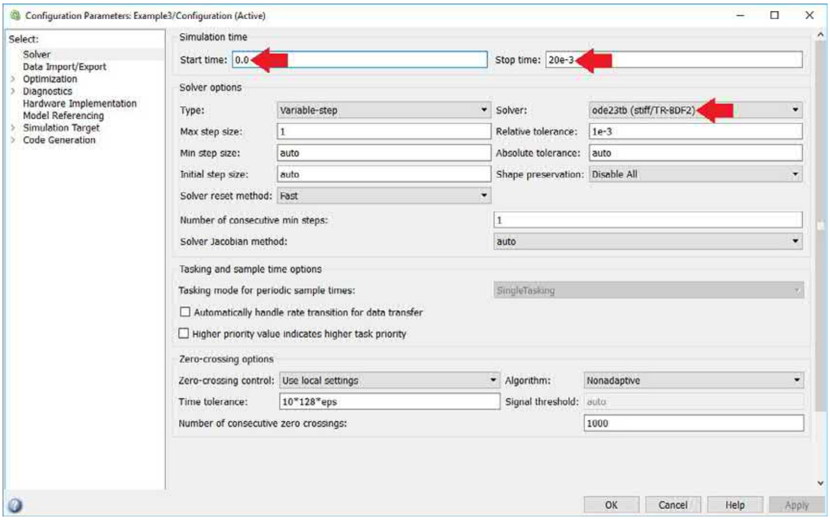

Example3 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example3
Example3
0.5
Constant
Duty
PLECS
Circuit
Vo
Circuit
Scope
Ready
100%
ode23tb

FIG. 4.149 The Simulink model.

The Simulink model configurations are shown in Fig. 4.150.


Configuration Parameters: Example3/Configuration (Active)
Select:
Solver
Data Import/Export
Optimization
Diagnostics
Hardware Implementation
Model Referencing
Simulation Target
Code Generation
OK
Cancel
Help
Apply

Simulation time

Start time: 0.0

Stop time: 20e-3

Solver options

Type: Variable-step

Solver: ode23tb (stiff/TR-BDF2)

Max step size: 1

Relative tolerance: 1e-3

Min step size: auto

Absolute tolerance: auto

Initial step size: auto

Shape preservation: Disable All

Solver reset method: Fast

Number of consecutive min steps: 1

Solver Jacobian method: auto

Tasking and sample time options

Tasking mode for periodic sample times: SingleTasking

☐ Automatically handle rate transition for data transfer

☐ Higher priority value indicates higher task priority

Zero-crossing options

Zero-crossing control: Use local settings

Algorithm: Nonadaptive

Time tolerance: 10*128*eps

Signal threshold: auto

Number of consecutive zero crossings: 1000

FIG. 4.150 Configuration Parameters window.

Fig. 4.151 shows the output voltage.

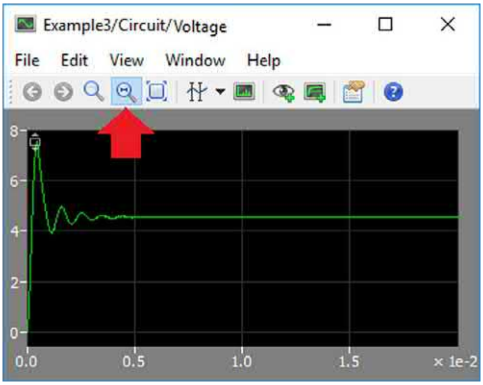

Example3/Circuit/Voltage
File
Edit
View
Window
Help
8
6
4
2
0
0.0
0.5
1.0
1.5
× 1e-2

FIG. 4.151 Output voltage (voltage of resistor).

You can use the magnifier icon (Fig. 4.151) to zoom into the steady-state portion of the graph and see the steady-state values. The steady-state portion of the graph is shown in Fig. 4.152.

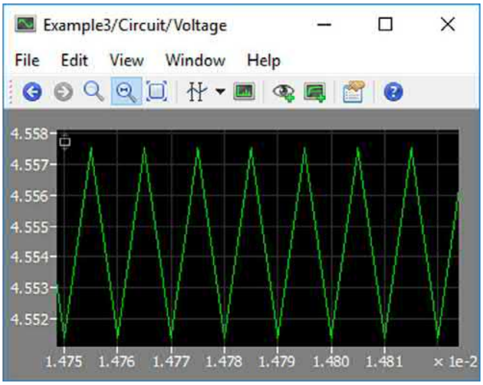

Example3/Circuit/Voltage
File
Edit
View
Window
Help
4.558
4.557
4.556
4.555
4.554
4.553
4.552
1.475
1.476
1.477
1.478
1.479
1.480
1.481
× 1e-2

FIG. 4.152 Steady-state portion of output voltage.

Fig. 4.153 shows the steady inductor current.

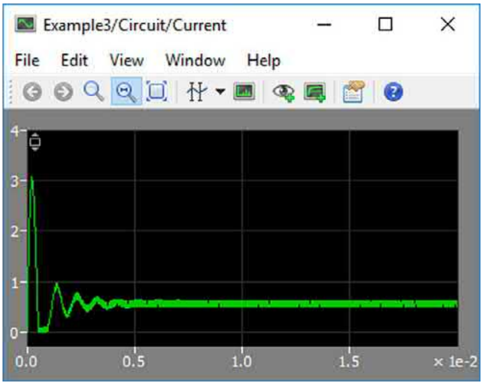

Example3/Circuit/Current
File
Edit
View
Window
Help
4
3
2
1
0
0.0
0.5
1.0
1.5
× 1e-2

FIG. 4.153 Inductor current.

The magnifier icon is used to see the steady-state portion of the graph.

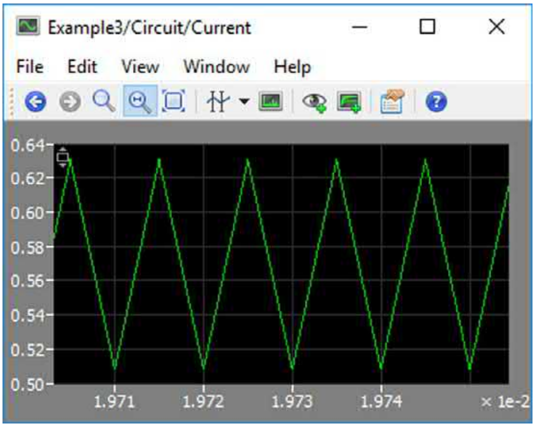

Example3/Circuit/Current
File Edit View Window Help
0.64
0.62
0.60
0.58
0.56
0.54
0.52
0.50
1.971
1.972
1.973
1.974
× 1e-2

FIG. 4.154 Steady-state portion of inductor current.

PLECS can calculate the steady-state operating point of converter by itself. So, there is no need to run the simulation and zoom into the steady-state portion of the graph. In order to calculate the Steady-state operating point, add a Steady-State Analysis block to the Simulink model as shown in Fig. 4.155. The Steady-State Analysis block can be found in the PLECS Extras section of Simulink Library Browser (Figs. 4.156 and 4.157).

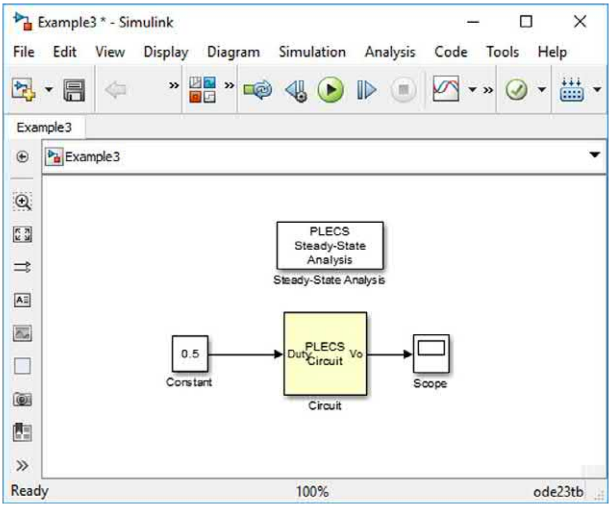

Example3 * - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools Help
Example3
Example3
PLECS Steady-State Analysis
Steady-State Analysis
0.5
Constant
Duty
PLECS Circuit
Vo
Circuit
Scope
Ready
100%
ode23tb

FIG. 4.155 Simulink model to extract the steady-state values.

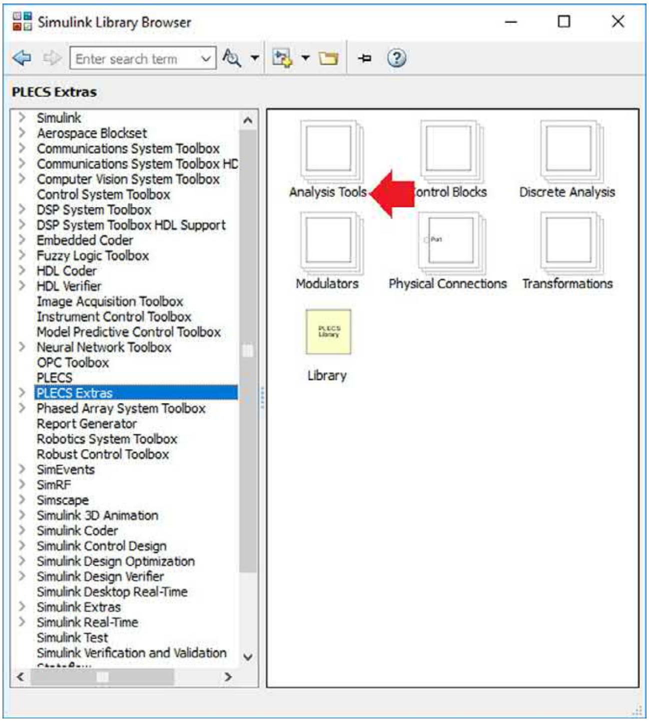

Simulink Library Browser
Enter search term
PLECS Extras
Simulink
Aerospace Blockset
Communications System Toolbox
Communications System Toolbox HD
Computer Vision System Toolbox
Control System Toolbox
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Desktop Real-Time
Simulink Extras
Simulink Real-Time
Simulink Test
Simulink Verification and Validation
Analysis Tools
ontrol Blocks
Discrete Analysis
Modulators
Physical Connections
Transformations
PLECS Library
Library

FIG. 4.156 Steady-State Analysis block can be found in the Analysis Tools section of Simulink Library Browser.

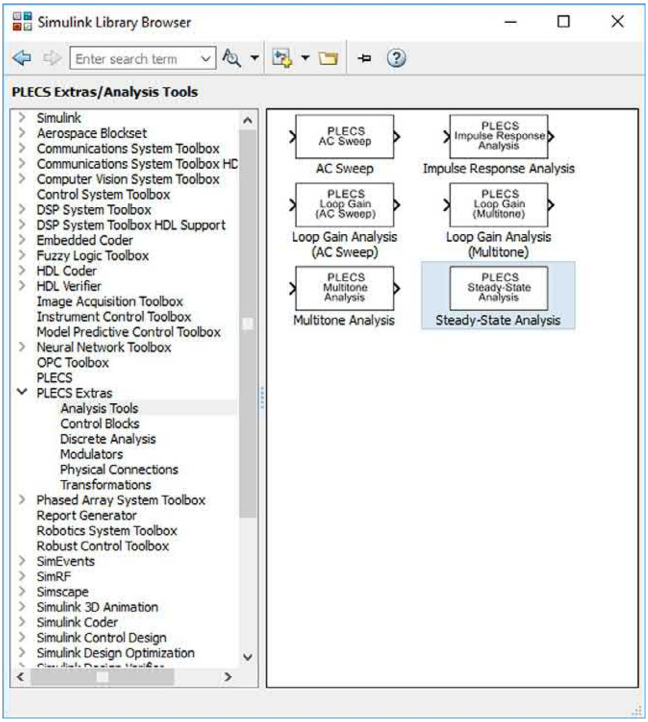
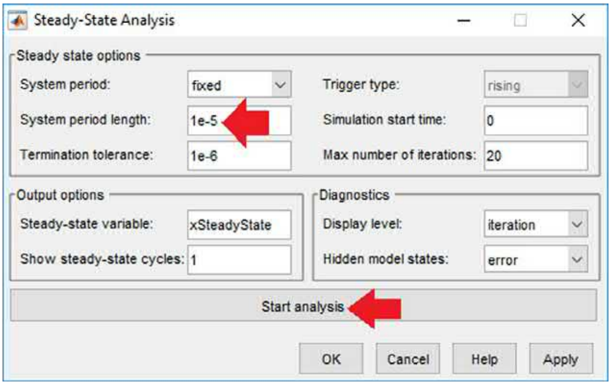

Simulink Library Browser
Enter search term
PLECS Extras/Analysis Tools
Simulink
Aerospace Blockset
Communications System Toolbox
Communications System Toolbox HD
Computer Vision System Toolbox
Control System Toolbox
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Analysis Tools
Control Blocks
Discrete Analysis
Modulators
Physical Connections
Transformations
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
PLECS AC Sweep
AC Sweep
PLECS Impulse Response Analysis
Impulse Response Analysis
PLECS Loop Gain (AC Sweep)
Loop Gain Analysis (AC Sweep)
PLECS Loop Gain (Multitone)
Loop Gain Analysis (Multitone)
PLECS Multitone Analysis
Multitone Analysis
PLECS Steady-State Analysis
Steady-State Analysis

FIG. 4.157 Steady-State Analysis block.

Double click the PLECS Steady-State Analysis block. The window shown in Fig. 4.158 will appear.


Steady-State Analysis
Steady state options
Output options
Diagnostics
Start analysis
OK
Cancel
Help
Apply

System period: fixed
System period length: 1e-5
Termination tolerance: 1e-6
Trigger type: rising
Simulation start time: 0
Max number of iterations: 20
Steady-state variable: xSteadyState
Show steady-state cycles: 1
Display level: iteration
Hidden model states: error

FIG. 4.158 The Steady-State Analysis block settings.

The System period length is filled with 1/f, where f is the switching frequency. Since the switching frequency is 100 kHz, it is filled with 1e − 5, which is equal to 10^{-5}. Click the Start analysis button (Fig. 4.158) to run the simulation. After simulation is finished, double click the PLECS circuit block in the Simulink model to open it. Double click the scopes to see the steady-state waveforms (Figs. 4.159 and 4.160). Compare these results with the result of previous analysis (Figs. 4.152 and 4.154). The results are the same.

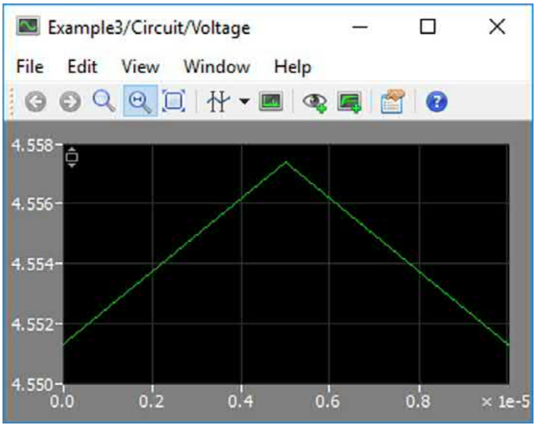

Example3/Circuit/Voltage
File
Edit
View
Window
Help
4.558
4.556
4.554
4.552
4.550
0.0
0.2
0.4
0.6
0.8
× 1e-5

FIG. 4.159 Steady-state output voltage waveform.

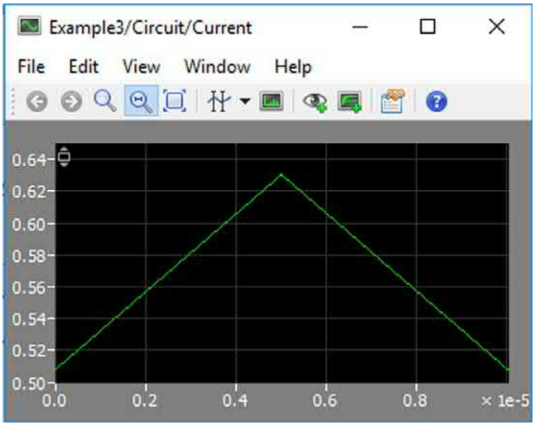

Example3/Circuit/Current
File
Edit
View
Window
Help
0.64
0.62
0.60
0.58
0.56
0.54
0.52
0.50
0.0
0.2
0.4
0.6
0.8
× 1e-5

FIG. 4.160 Steady-state inductor current waveform.

4.13 More simulation examples

Simulink version of PLECS comes with many different simulation examples. To access this example, double click the Demos icon in the Simulink Library Browser (Fig. 4.161).

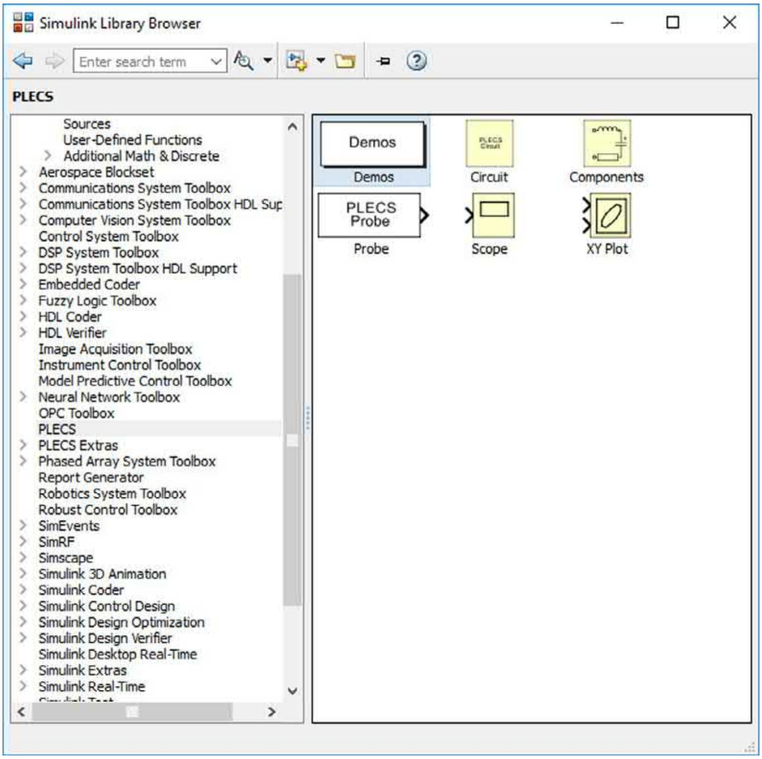
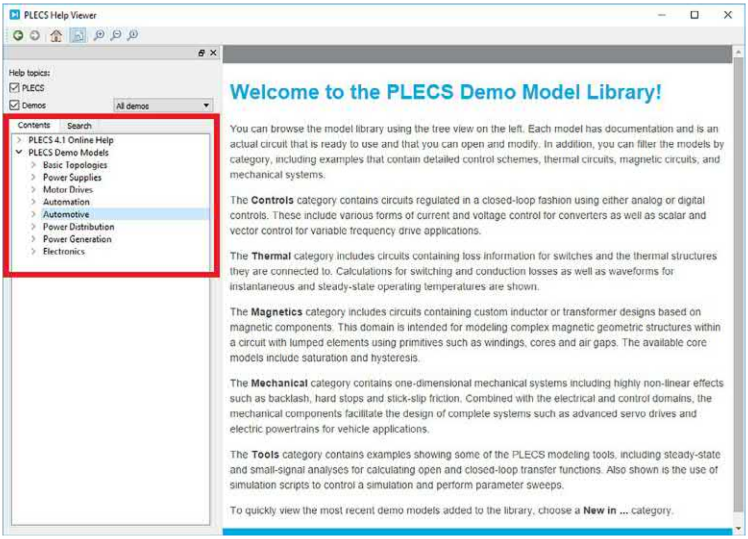

Simulink Library Browser
Enter search term
PLECS
Sources
User-Defined Functions
Additional Math & Discrete
Aerospace Blockset
Communications System Toolbox
Communications System Toolbox HDL Sup
Computer Vision System Toolbox
Control System Toolbox
DSP System Toolbox
DSP System Toolbox HDL Support
Embedded Coder
Fuzzy Logic Toolbox
HDL Coder
HDL Verifier
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
PLECS
PLECS Extras
Phased Array System Toolbox
Report Generator
Robotics System Toolbox
Robust Control Toolbox
SimEvents
SimRF
Simscape
Simulink 3D Animation
Simulink Coder
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Desktop Real-Time
Simulink Extras
Simulink Real-Time
Demos
Demos
Circuit
Components
PLECS Probe
Probe
Scope
XY Plot

FIG. 4.161 Demos section of PLECS block set.

After you double click the Demos, PLECS Help Viewer will be opened. Use the Contents tab in the left of window to select the category you want (Fig. 4.162).


PLECS Help Viewer
Help topics:
☑ PLECS
☑ Demos
All demos
Contents
Search
PLECS 4.1 Online Help
PLECS Demo Models
Basic Topologies
Power Supplies
Motor Drives
Automation
Automotive
Power Distribution
Power Generation
Electronics

Welcome to the PLECS Demo Model Library!

You can browse the model library using the tree view on the left. Each model has documentation and is an actual circuit that is ready to use and that you can open and modify. In addition, you can filter the models by category, including examples that contain detailed control schemes, thermal circuits, magnetic circuits, and mechanical systems.

The **Controls** category contains circuits regulated in a closed-loop fashion using either analog or digital controls. These include various forms of current and voltage control for converters as well as scalar and vector control for variable frequency drive applications.

The **Thermal** category includes circuits containing loss information for switches and the thermal structures they are connected to. Calculations for switching and conduction losses as well as waveforms for instantaneous and steady-state operating temperatures are shown.

The **Magnetics** category includes circuits containing custom inductor or transformer designs based on magnetic components. This domain is intended for modeling complex magnetic geometric structures within a circuit with lumped elements using primitives such as windings, cores and air gaps. The available core models include saturation and hysteresis.

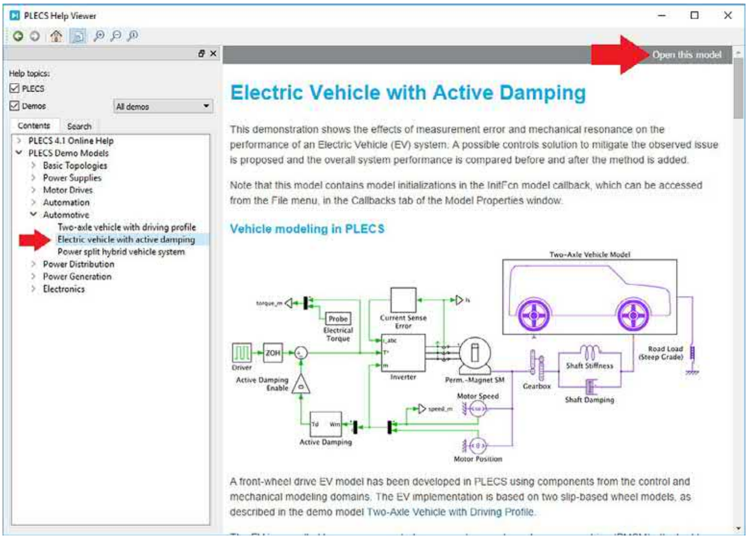
The **Mechanical** category contains one-dimensional mechanical systems including highly non-linear effects such as backlash, hard stops and stick-slip friction. Combined with the electrical and control domains, the mechanical components facilitate the design of complete systems such as advanced servo drives and electric powertrains for vehicle applications.

The **Tools** category contains examples showing some of the PLECS modeling tools, including steady-state and small-signal analyses for calculating open and closed-loop transfer functions. Also shown is the use of simulation scripts to control a simulation and perform parameter sweeps.

To quickly view the most recent demo models added to the library, choose a **New in ...** category.

FIG. 4.162 PLECS Help Viewer.

For instance, assume we want to see prepared examples related to Automotive applications. We click the Automotive section of Contents. A description about the simulation appears in the right window. You can open the model by clicking the Open this model (Fig. 4.163).


PLECS Help Viewer
Open this model
Help topics:
☑ PLECS
☑ Demos
All demos
Contents
Search
PLECS 4.1 Online Help
PLECS Demo Models
Basic Topologies
Power Supplies
Motor Drives
Automation
Automotive
Two-axle vehicle with driving profile
Electric vehicle with active damping
Power split hybrid vehicle system
Power Distribution
Power Generation
Electronics
Two-Axle Vehicle Model
torque_m
Probe
Electrical Torque
Current Sense Error
is
i_abc
T*
m
Driver
ZOH
Inverter
Perm.-Magnet SM
Gearbox
Shaft Stiffness
Shaft Damping
Road Load (Steep Grade)
Active Damping Enable
Motor Speed
speed_m
Td
Wm
Active Damping
Motor Position

Electric Vehicle with Active Damping

This demonstration shows the effects of measurement error and mechanical resonance on the performance of an Electric Vehicle (EV) system. A possible controls solution to mitigate the observed issue is proposed and the overall system performance is compared before and after the method is added.

Note that this model contains model initializations in the InitFcn model callback, which can be accessed from the File menu, in the Callbacks tab of the Model Properties window.

Vehicle modeling in PLECS

A front-wheel drive EV model has been developed in PLECS using components from the control and mechanical modeling domains. The EV implementation is based on two slip-based wheel models, as described in the demo model Two-Axle Vehicle with Driving Profile.

FIG. 4.163 A sample example related to automotive. Click the Open this model to open the prepared simulation in Simulink.

After you clicked the Open this model, the prepared simulation will appear in the Simulink environment (Fig. 4.164).

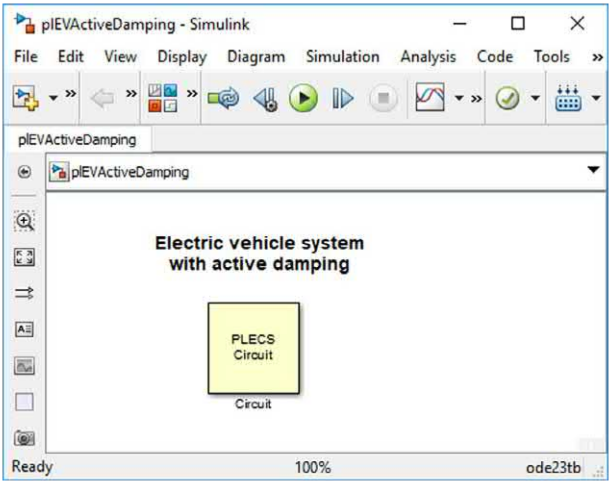

pIEVActiveDamping - Simulink
File Edit View Display Diagram Simulation Analysis Code Tools »
pIEVActiveDamping
pIEVActiveDamping
Electric vehicle system
with active damping
PLECS
Circuit
Circuit
Ready
100%
ode23tb

FIG. 4.164 After Open this model is clicked, the prepared sample simulation is opened in the Simulink environment.

Further reading

[1] F. Asadi, N. Abut, Simulation of power electronics converters with MATLAB/Simulink® (in Turkish), Umuttepe yayınları, 2018.

[2] F. Asadi, K. Eguchi, Dynamics and control of DC-DC converters, Morgan & Claypool, 2018. ISBN: 978-168-1732-98-5.

[3] F. Asadi, Computer techniques for dynamic modelling of DC-DC power converters, Morgan & Claypool, 2018. ISBN: 978-168-1734-19-4.

[4] F. Asadi, N. Abut, et al., Development of a power electronics converter dynamics toolbox for MATLAB, Int. J. Adv. Appl. Sci. 4 (6) (2017) 56–62.

[5] F. Asadi, N. Abut, et al., Designing a PI controller for Cuk converter using converter dynamics toolbox for MATLAB, Int. J. Adv. Appl. Sci. 4 (6) (2017) 175–180.

[6] F. Asadi, N. Abut, et al., Joy of controller design: controller design based on Kocaeli University's converter dynamics toolbox for MATLAB, Int. J. Adv. Appl. Sci. 4 (7) (2017) 5–10.

[7] F. Asadi, N. Abut, et al., KUCA: Kocaeli University Converter Analysis simulation software in power electronics, Int. J. Adv. Appl. Sci. 3 (12) (2016) 55–61.

[8] F. Asadi, N. Abut, et al., Pole placement based on derivative of states, Int. J. Adv. Appl. Sci. 3 (10) (2016) 100–102.

Chapter 5

Thermal analysis of power electronics converters with PLECS

Chapter outline

5.1 Introduction 299
5.2 Single-phase open-loop inverter 300
5.3 Electrical simulation of single-phase inverter 302
5.4 Thermal description of semiconductor switches 311
5.5 Switching losses 311
  5.5.1 Turn-on switching losses for the IGBT 312
  5.5.2 Turn-off switching losses for the IGBT 317
5.6 Conduction losses for the IGBT 320
5.7 Thermal impedances 321
5.8 Adding comments 324
5.9 Saving the produced thermal model 325
5.10 Adding the produced model to thermal search path of PLECS 326
5.11 Modeling losses of body diode 328
  5.11.1 The turn-on losses of body diode 330
  5.11.2 The turn-off losses of body diode 330
  5.11.3 The conduction losses of body diode 331
5.12 Thermal impedance of body diode 331
5.13 Loss measurements 335
  5.13.1 Calculation of IGBT's losses 337
  5.13.2 Calculation of body diode's losses 342
  5.13.3 Calculation of total losses 342
5.14 Junction temperatures measurement 348
5.15 Running the simulation 349
5.16 Designing the heat sink 352
5.17 Effect of modulation technique on losses 358
  5.17.1 Review of unipolar PWM 358
5.18 Calculation of losses for a unipolar PWM inverter 360
Further reading 363

5.1 Introduction

The previous chapters focused on the electrical simulations of power electronics circuits. This chapter focuses on the thermal aspects of power electronics converters. Power stage of a power electronics converter is composed of capacitors, inductors, and semiconductors switches. Semiconductor switches dissipate a

Simulation of Power Electronics Converters Using PLECS®. https://doi.org/10.1016/B978-0-12-817364-0.00005-9
© 2020 Elsevier Inc. All rights reserved.

part of energy in form of heat. The energy losses in the semiconductor switches increase the temperature of device. The produced thermal energy must be taken from the semiconductor switch with the aid of a heat sink; otherwise, the semiconductor switch fails, i.e., it burns. So, good thermal design, i.e., designing a suitable heat sink, is an important part of power electronics converters design process.

This chapter focuses on the thermal analysis capabilities of PLECS®. A single-phase inverter is studied as an illustrative example in this chapter. The switches energy losses are modeled with the aid of datasheet. Total energy losses are calculated with PLECS® and a suitable heat sink is designed for the inverter. Effect of modulation strategy on inverter losses is studied as well. Thermal analysis of other types of power electronics converters can be done in the same way shown in this chapter.

5.2 Single-phase open-loop inverter

Single-phase inverters are commonly used for residential photovoltaic applications, uninterruptible power supplies, and standalone power supplies. One important design aspect for an inverter is the thermal design. In particular, the heat sink thermal resistance must be calculated in order to keep the peak heat sink and junction temperatures within specified design limits. The thermal design is highly dependent on the semiconductor losses, which can be calculated easily using the thermal modeling features of PLECS.

Consider the single-phase inverter shown in Fig. 5.1.

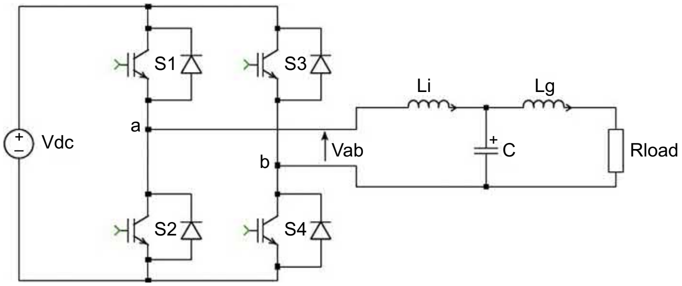


FIG. 5.1 The schematic of single-phase inverter. Assume that inverter legs (i.e., switch S1-S2 or S3-S4) are realized with Infineon EasyPACK™ IGBT Power Modules F4-50R06W1E3. The datasheet can be found in appendixes. According to datasheet, the device junction can withstand up to 175°C; however, in this chapter, we limit the maximum junction temperature to 125°C in order to increase the device life.

The inverter parameters are shown in Table 5.1.

TABLE 5.1 Parameters of single-phase inverter shown in Fig. 5.1.

Input DC voltage	400 V
Peak output voltage	325 V
Output frequency	50 Hz
Load resistance	17.6 Ω
Switching frequency	16 kHz
L_i	0.75 mH
L_g	0.5 mH
C	2.2 μF

We use the open loop control to the output voltage in this example. Block diagram of open-loop voltage control strategy is given in Fig. 5.2. We want to obtain a sinusoidal output voltage with peak amplitude of 325 V.

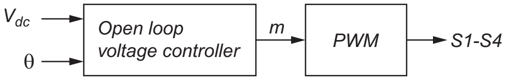


FIG. 5.2 Open-loop control of single-phase inverter.

The switches are modulated using a bipolar sinusoidal PWM strategy as depicted in Fig. 5.3. With bipolar modulation, the switches S1, S4 are turned on together as a pair to create a positive voltage at the inverter output terminals and the switch pair S2, S3 are turned on to create a negative output voltage.

The modulation index is calculated in order to create a sinusoidal output voltage with a peak amplitude of 325 V.

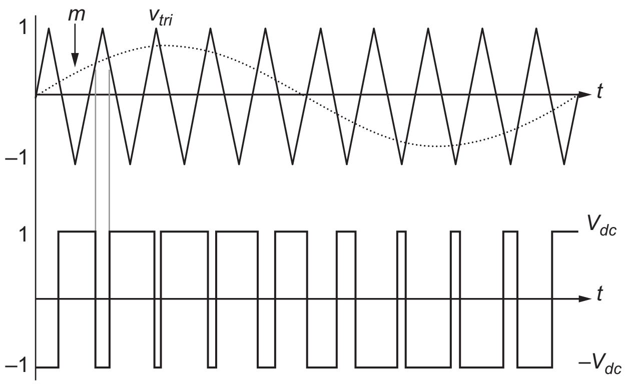


FIG. 5.3 Bipolar modulation.

$$m = \frac{V_O{}^*}{V_{dc}} = \frac{325\sin(\theta)}{V_{dc}},$$

where $V_O{}^*$ is the reference sinusoidal output voltage and $\theta = 2\pi ft$ is the reference output voltage angle given as a function of output frequency, f, and time, t.

5.3 Electrical simulation of single-phase inverter

Fig. 5.4 shows the schematic of single-phase inverter in the PLECS environment. We want to obtain the electrical waveforms of the circuit in this section.

Settings of blocks used are shown in Figs. 5.5–5.7.

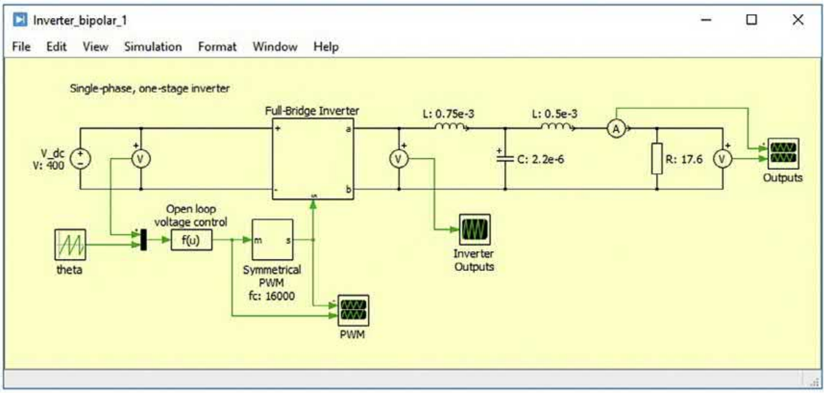
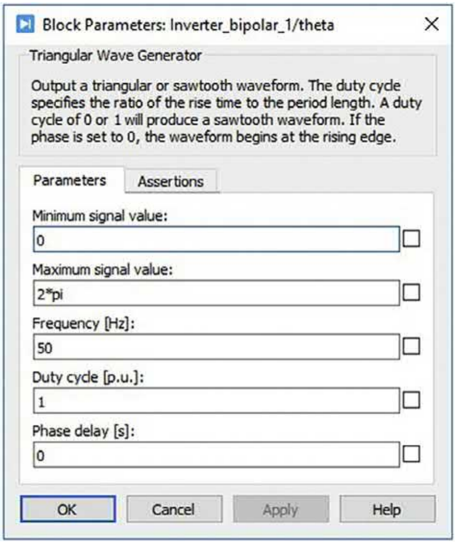

Inverter_bipolar_1
File Edit View Simulation Format Window Help
Single-phase, one-stage inverter
Full-Bridge Inverter
a
b
L: 0.75e-3
L: 0.5e-3
A
C: 2.2e-6
R: 17.6
V
V_dc
V: 400
Outputs
Open loop
voltage control
f(u)
m
s
theta
Symmetrical
PWM
fc: 16000
Inverter
Outputs
PWM

FIG. 5.4 Schematic of single-phase inverter in the PLECS environment.


Block Parameters: Inverter_bipolar_1/theta
Parameters
Assertions
OK
Cancel
Apply
Help

Triangular Wave Generator

Output a triangular or sawtooth waveform. The duty cycle specifies the ratio of the rise time to the period length. A duty cycle of 0 or 1 will produce a sawtooth waveform. If the phase is set to 0, the waveform begins at the rising edge.

Minimum signal value: 0 ☐

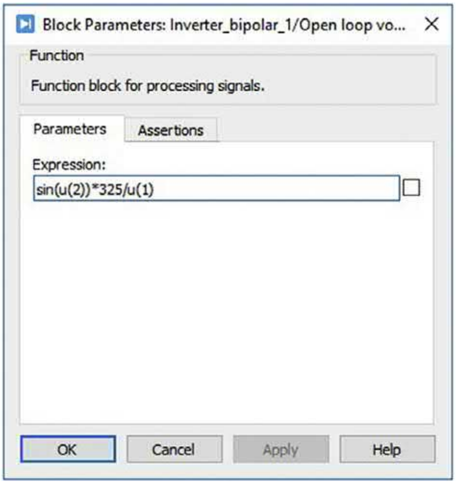
Maximum signal value: 2*pi ☐

Frequency [Hz]: 50 ☐

Duty cycle [p.u.]: 1 ☐

Phase delay [s]: 0 ☐

FIG. 5.5 Settings of theta block.



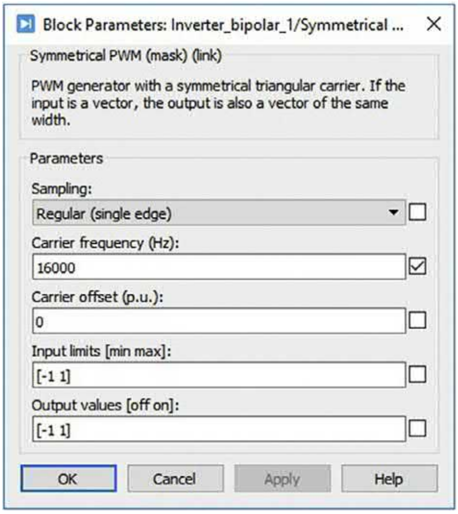
Block Parameters: Inverter_bipolar_1/Open loop vo...

Function

Function block for processing signals.

Expression: sin(u(2))*325/u(1) ☐

FIG. 5.6 Setting of Open-loop voltage control Function block.



Block Parameters: Inverter_bipolar_1/Symmetrical ...

Symmetrical PWM (mask) (link)

PWM generator with a symmetrical triangular carrier. If the input is a vector, the output is also a vector of the same width.

Parameters

Sampling: Regular (single edge) ☐

Carrier frequency (Hz): 16000 ☑

Carrier offset (p.u.): 0 ☐

Input limits [min max]: [-1 1] ☐

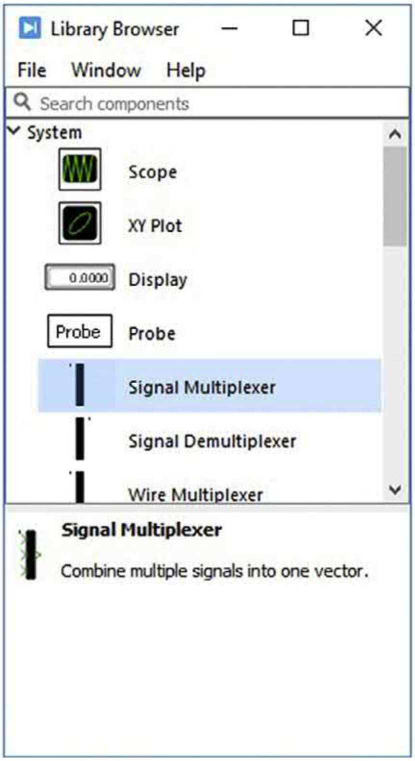
Output values [off on]: [-1 1] ☐

FIG. 5.7 Settings of Symmetrical PWM block.

Value of input DC voltage (V_{dc}) and θ is entered into the Open-loop voltage control block with the aid of a Signal Multiplexer block (Fig. 5.8).

Inside of Full-bridge Inverter subsystem is shown in Fig. 5.9. The two function blocks (Fig. 5.10) close the required switches according to the signal entering s port of subsystem. IGBT1 and IGBT4 are closed when the signal entered to s port is positive, and IGBT2 and IGBT3 are closed when the signal entered s port is negative.

Figs. 5.11 and 5.12 show the settings of the two function blocks.


Library Browser
File
Window
Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer

Signal Multiplexer

Combine multiple signals into one vector.

FIG. 5.8 Signal Multiplexer block can be found in the System section of Library Browser.

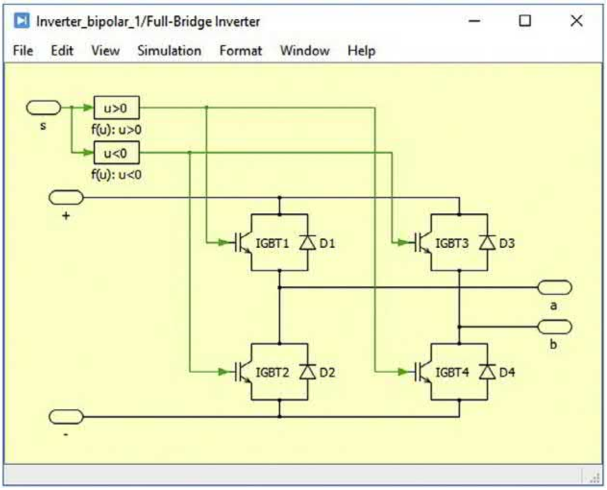
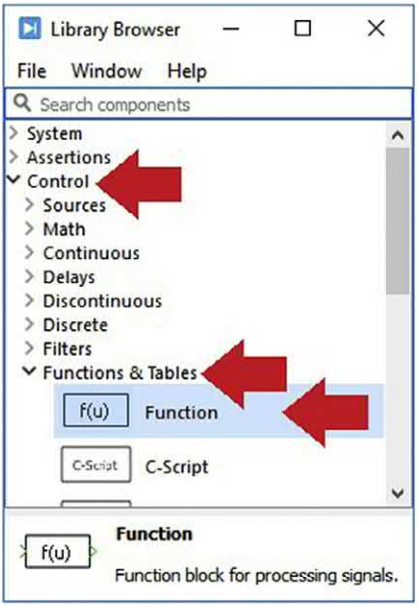

Inverter_bipolar_1/Full-Bridge Inverter
File
Edit
View
Simulation
Format
Window
Help
s
u>0
f(u): u>0
u<0
f(u): u<0
+
IGBT1
D1
IGBT3
D3
a
b
IGBT2
D2
IGBT4
D4
-

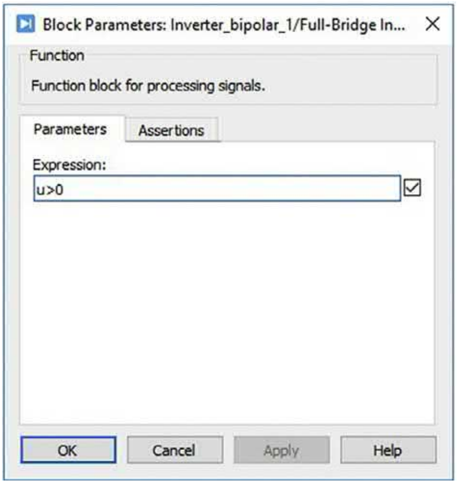
FIG. 5.9 Inside the Full-Bridge Inverter subsystem.


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
f(u)
Function
C-Script
C-Script
f(u)

Function

Function block for processing signals.

FIG. 5.10 Function block can be found in the functions and tables section of Library Browser.

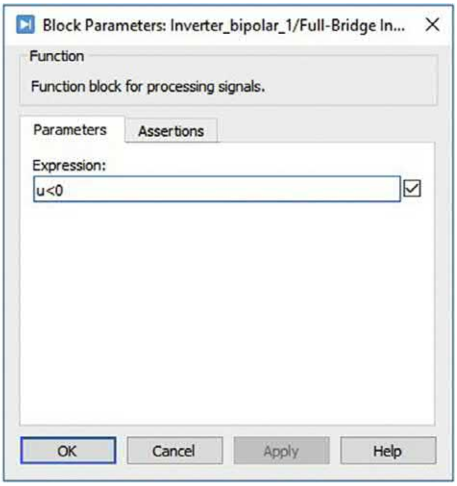

Block Parameters: Inverter_bipolar_1/Full-Bridge In...
Parameters
Assertions
OK
Cancel
Apply
Help

Function

Function block for processing signals.

Expression: u>0 ☑

FIG. 5.11 Settings of upper Function block (see Fig. 5.9).



Block Parameters: Inverter_bipolar_1/Full-Bridge In...

Function

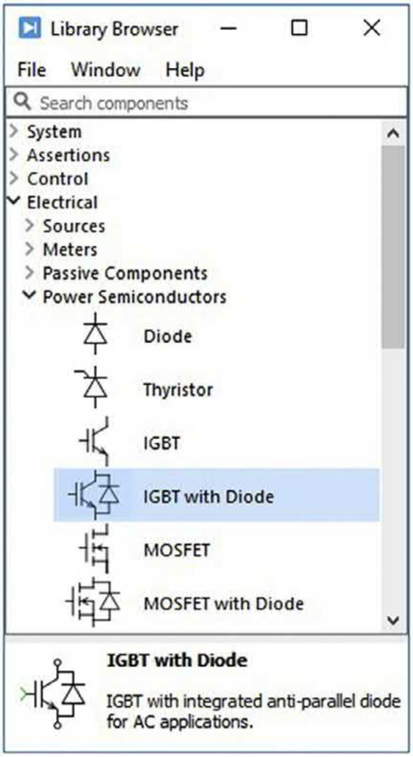
Function block for processing signals.

Expression: u<0 ☑

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 5.12 Setting of lower Function block (see Fig. 5.9).

Don’t use the IGBT with diode block (Fig. 5.13) for the power stage of inverter. Instead use the IGBT and diode blocks (Fig. 5.14). This simplifies the thermal modeling of converter.



Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Diode
Thyristor
IGBT
IGBT with Diode
MOSFET
MOSFET with Diode

IGBT with Diode

IGBT with integrated anti-parallel diode for AC applications.

FIG. 5.13 IGBT with Diode block.

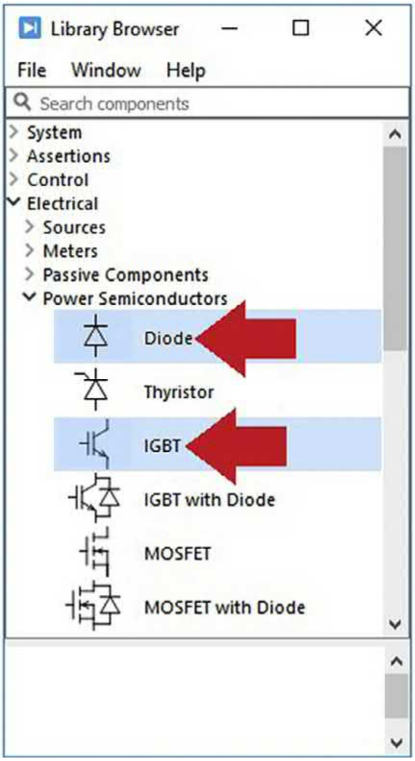
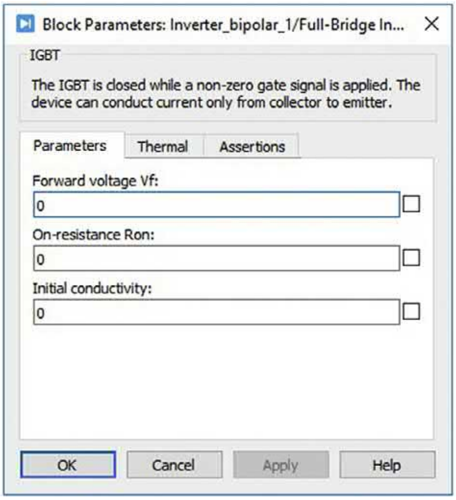

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Meters
Passive Components
Power Semiconductors
Diode
Thyristor
IGBT
IGBT with Diode
MOSFET
MOSFET with Diode

FIG. 5.14 IGBT and Diode blocks.

Settings of IGBT's and diodes are shown in Figs. 5.15 and 5.16, respectively.


Block Parameters: Inverter_bipolar_1/Full-Bridge In...

IGBT

The IGBT is closed while a non-zero gate signal is applied. The device can conduct current only from collector to emitter.

Parameters
Thermal
Assertions

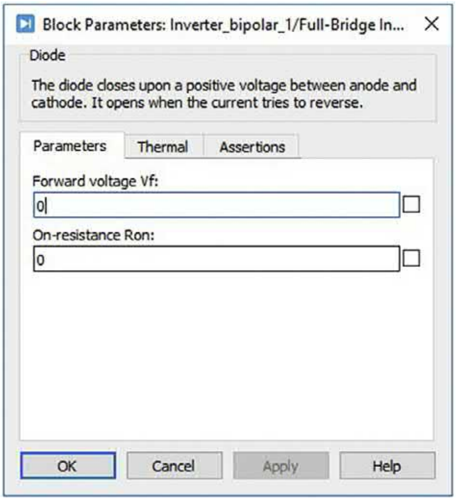
Forward voltage Vf: 0 ☐

On-resistance Ron: 0 ☐

Initial conductivity: 0 ☐

OK
Cancel
Apply
Help

FIG. 5.15 IGBT block settings. All the IGBTs have these settings.


Block Parameters: Inverter_bipolar_1/Full-Bridge In...

Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

Forward voltage Vf: 0

On-resistance Ron: 0

Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

FIG. 5.16 Diode block settings. All the diodes have these settings.

Click the Simulation parameters (Fig. 5.17) and change the settings as shown in Fig. 5.18.

Press Ctrl+T or click the Start (Fig. 5.19) to run the simulation.

The simulation results are shown in Figs. 5.20–5.24.

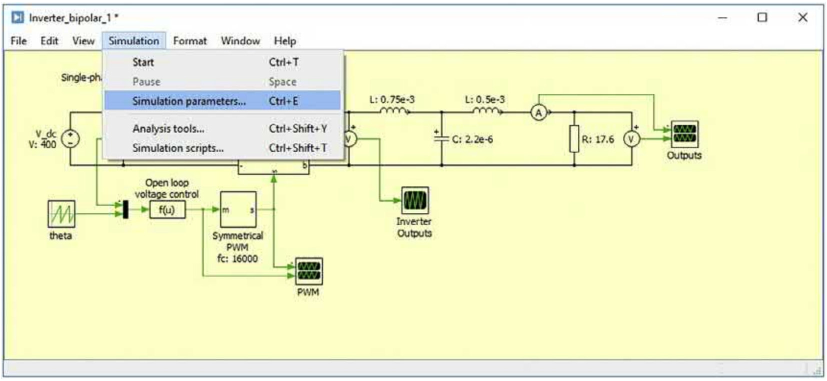
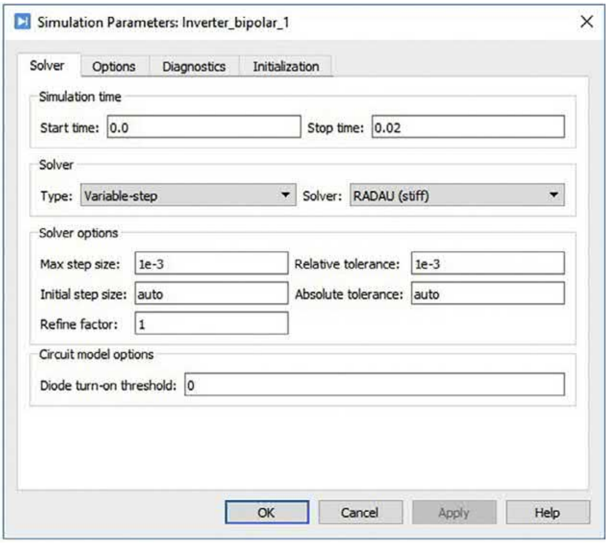

Inverter_bipolar_1 *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
Single-ph
V_dc
V: 400
L: 0.75e-3
L: 0.5e-3
C: 2.2e-6
R: 17.6
A
V
Outputs
Inverter Outputs
theta
Open loop voltage control
f(u)
m
s
Symmetrical PWM
fc: 16000
PWM

FIG. 5.17 Simulation Parameters window can be made visible with the aid of Simulation parameters in the Simulation window.



Simulation Parameters: Inverter_bipolar_1

Solver
Options
Diagnostics
Initialization

Simulation time

Start time: 0.0

Stop time: 0.02

Solver

Type: Variable-step

Solver: RADAU (stiff)

Solver options

Max step size: 1e-3

Relative tolerance: 1e-3

Initial step size: auto

Absolute tolerance: auto

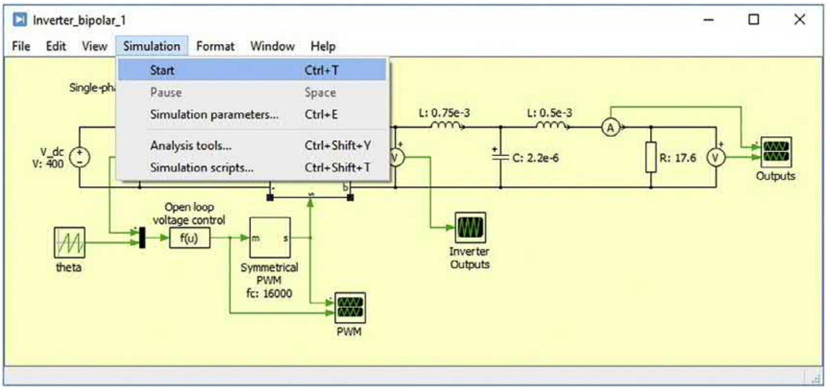
Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

OK
Cancel
Apply
Help

FIG. 5.18 Simulation Parameters window.



Inverter_bipolar_1
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
Single-ph
V_dc
V: 400
L: 0.75e-3
L: 0.5e-3
C: 2.2e-6
R: 17.6
A
V
V
Outputs
Inverter
Outputs
Open loop
voltage control
f(u)
theta
m
s
Symmetrical
PWM
fc: 16000
PWM

FIG. 5.19 Running the simulation.

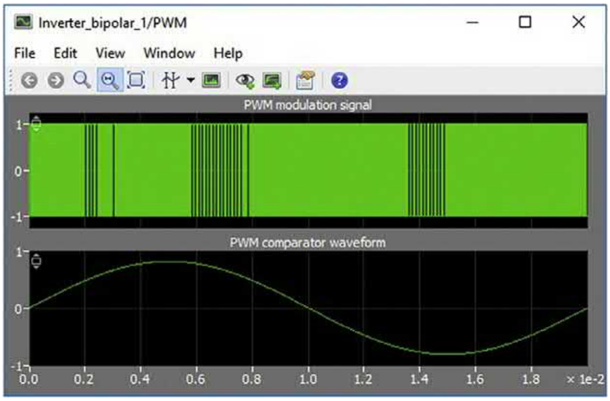


FIG. 5.20 PWM scope waveforms. Use the magnifier icon to zoom in.

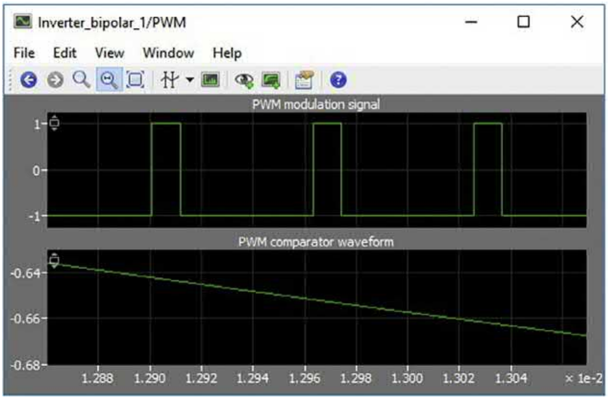

Inverter_bipolar_1/PWM
File Edit View Window Help
PWM modulation signal
1
0
-1
PWM comparator waveform
-0.64
-0.66
-0.68
1.288 1.290 1.292 1.294 1.296 1.298 1.300 1.302 1.304
× 1e-2

FIG. 5.21 Close up of PWM scope waveforms.

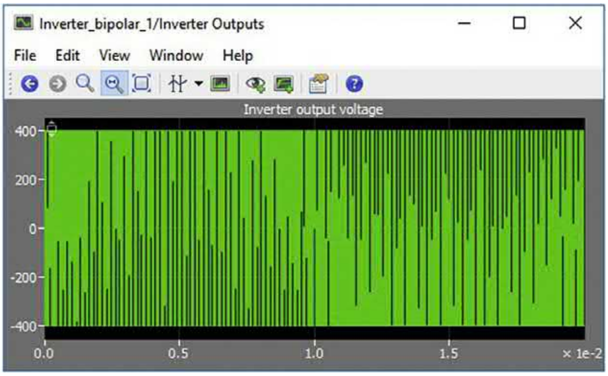

Inverter_bipolar_1/Inverter Outputs
File Edit View Window Help
Inverter output voltage
400
200
0
-200
-400
0.0 0.5 1.0 1.5
× 1e-2

FIG. 5.22 Inverter Outputs scope waveform.

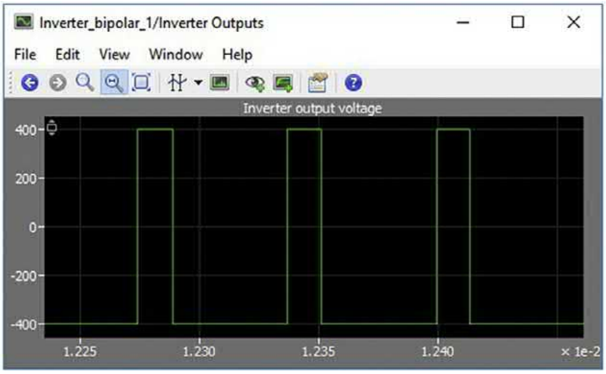

Inverter_bipolar_1/Inverter Outputs
File Edit View Window Help
Inverter output voltage
400
200
0
-200
-400
1.225 1.230 1.235 1.240
× 1e-2

FIG. 5.23 Close up of Inverter output voltage scope waveform.

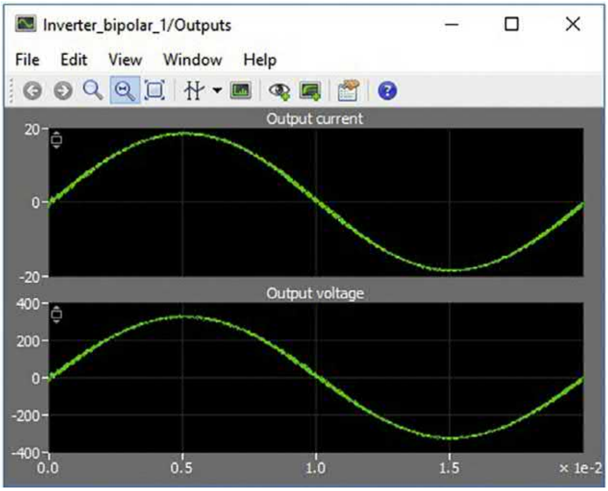

Inverter_bipolar_1/Outputs
File
Edit
View
Window
Help
Output current
20
0
-20
Output voltage
400
200
0
-200
-400
0.0
0.5
1.0
1.5
× 1e-2

FIG. 5.24 Outputs scope waveform.

5.4 Thermal description of semiconductor switches

To calculate the inverter losses and simulate the thermal performance of the inverter, thermal loss descriptions of the semiconductors are first required. Using the data sheet for the Infineon F4-50R06W1E3 IGBT module, we will create a thermal description for the IGBT and body diode. We will need to enter the following information for each component: conduction losses, switching losses, and thermal impedance between junction and case.

5.5 Switching losses

The current and voltage of a semiconductor switch does not change instantaneously. This concept is shown in Fig. 5.25.

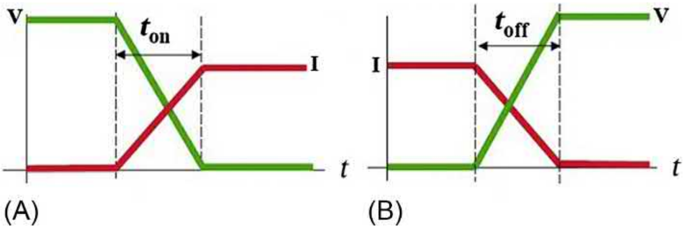

V
t_{on}
I
t
(A)
t_{off}
V
I
t
(B)

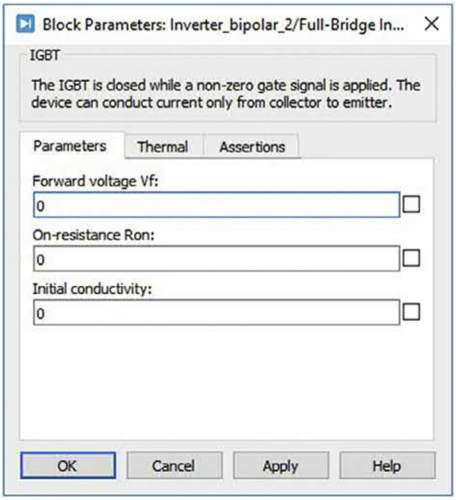
FIG. 5.25 The semiconductor switch (A) turns on (B) turns off.

As shown in Fig. 5.25A and B, during the t_{off} or t_{on} the current and voltage are not zero, so their product, i.e., the power loss, is not zero. The power lost during the turn on is called turn-on switching loss and the power during the turn off is called turn-off switching loss. Switching loss happens only in transitions, i.e., when an on device turns off or when an off device turns on.

The switching losses are modeled (with the aid of datasheet) in the next sections.

5.5.1 Turn-on switching losses for the IGBT

In order to define the switching losses of IGBT, open the Full-Bridge Inverter subsystem and double click on the IGBTs, i.e., IGBT1. A window like the one shown in Fig. 5.26 will be opened.



Block Parameters: Inverter_bipolar_2/Full-Bridge In...

IGBT

The IGBT is closed while a non-zero gate signal is applied. The device can conduct current only from collector to emitter.

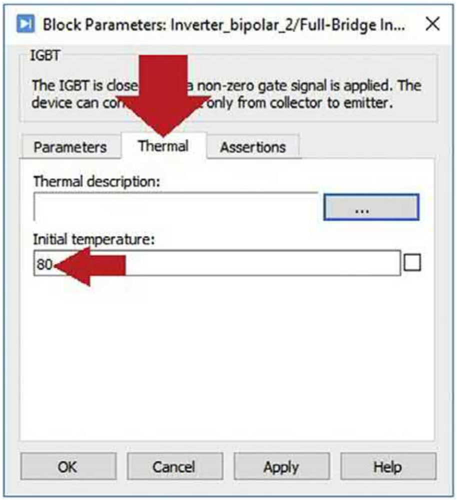
Forward voltage Vf: 0 ☐

On-resistance Ron: 0 ☐

Initial conductivity: 0 ☐

FIG. 5.26 IGBT parameters window.

Click the Thermal tab (Fig. 5.27).



Block Parameters: Inverter_bipolar_2/Full-Bridge In...

IGBT

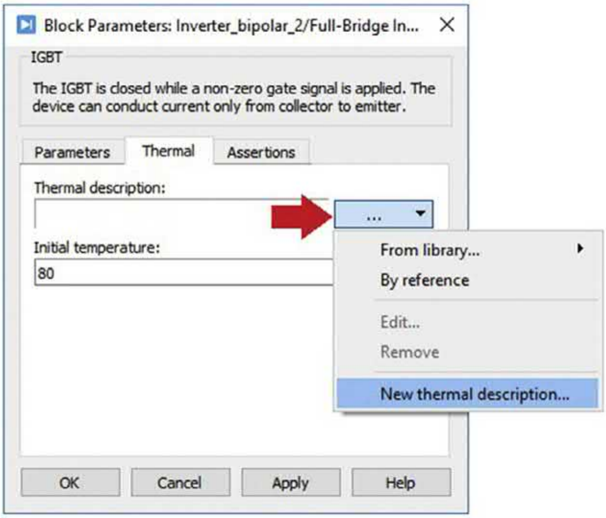
The IGBT is close… a non-zero gate signal is applied. The device can co… only from collector to emitter.

Thermal description:

Initial temperature: 80 ☐

FIG. 5.27 Thermal tab of IGBT. It is assumed that junction initial temperature is 80°C.

Click the button shown in Fig. 5.28 to open the drop down list. Click the New thermal description to enter the switching and conduction losses of IGBT.

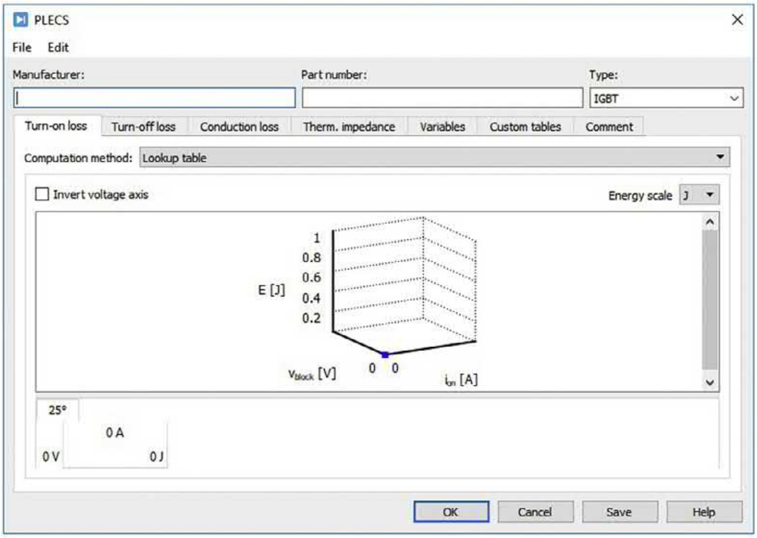

Block Parameters: Inverter_bipolar_2/Full-Bridge In...
IGBT
Parameters
Thermal
Assertions
Thermal description:
...
From library...
By reference
Edit...
Remove
New thermal description...
OK
Cancel
Apply
Help

The IGBT is closed while a non-zero gate signal is applied. The device can conduct current only from collector to emitter.

Initial temperature: 80

FIG. 5.28 The thermal behavior of IGBT is defined with the aid of New thermal description.

After you clicked the New thermal description, the window shown in Fig. 5.29 will appear. The switching and conduction losses of IGBT are set with the aid of this window.


PLECS
File
Edit
Manufacturer:
Part number:
Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
Invert voltage axis
E [J]
1
0.8
0.6
0.4
0.2
V_{block} [V]
0
0
i_{on} [A]
25°
0 A
0 V
0 J
OK
Cancel
Save
Help

Type: IGBT

Computation method: Lookup table

☐ Invert voltage axis

Energy scale J

FIG. 5.29 It is recommended to click the Help button and read the provided help page of window in order to see the details of this window.

First of all enter the manufacturer name and part number to the Manufacturer and Part number boxes, respectively. Set the Type to IGBT since we are modeling an IGBT (Fig. 5.30).

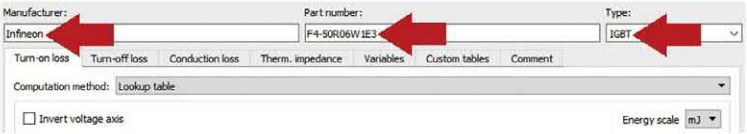

Manufacturer: Infineon
Part number: F4-50R06W1E3
Type: IGBT
Computation method: Lookup table
☐ Invert voltage axis
Energy scale mJ

FIG. 5.30 Entering the IGBT information.

Set the Computation method to Lookup table. Selection of Lookup table lets you enter the blocking voltage, the device current, and the junction temperature according to the graphs given in datasheet. Set the Energy scale to milli Joul (mj), since the datasheets generally give the losses in terms of mJ (Fig. 5.31).

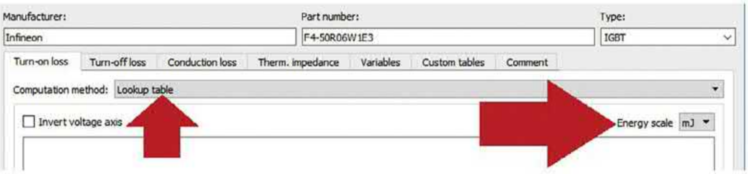

Manufacturer: Infineon
Part number: F4-50R06W1E3
Type: IGBT
Computation method: Lookup table
☐ Invert voltage axis
Energy scale mJ

FIG. 5.31 It is recommended to select Lookup table for Computation method and mj for Energy scale. When you use Lookup table, PLECS uses interpolation (or extrapolation) to calculate values that are not defined in the given table.

The graph of switching losses for IGBT is shown in Fig. 5.32. The datasheet gives the switching losses for two junction temperatures, 125°C and 150°C. We use the E_{on} curves to model the turn on switching losses of the IGBT.

Go to Turn-on loss tab in order to enter the turn-on losses (Fig. 5.33).

Enter the data for 125°C and 150°C to the software (Figs. 5.34 and 5.35). The datasheet only provides the data for 300 V. To add a linear relationship between the switching losses and voltage, extend loss line to 0 V by entering loss values 0 mJ at 0 V for all currents.

You can add a new temperature tab to the table by right clicking on the temperature tab and selecting the New temperature values (Fig. 5.36).

You can add a new current column to the table by right clicking on one of the current columns and selecting the New current values (Fig. 5.37).

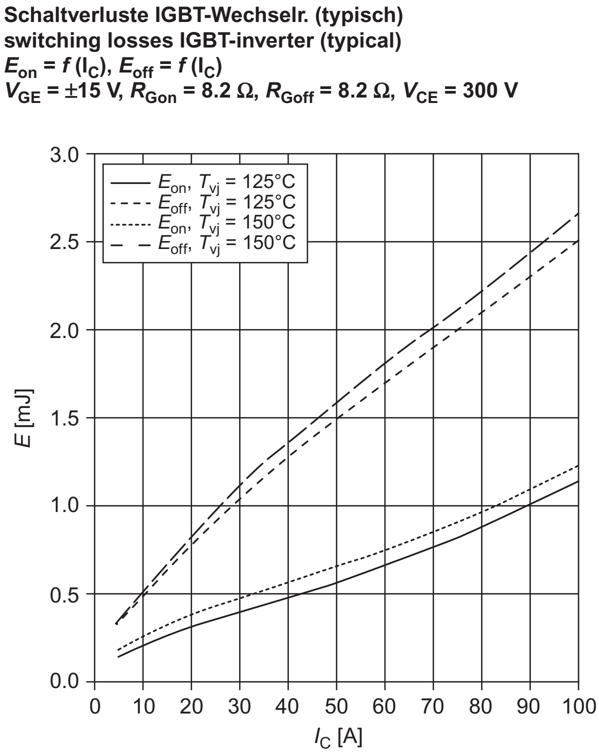
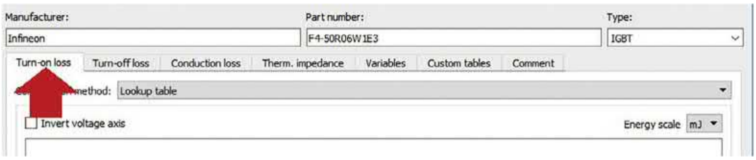


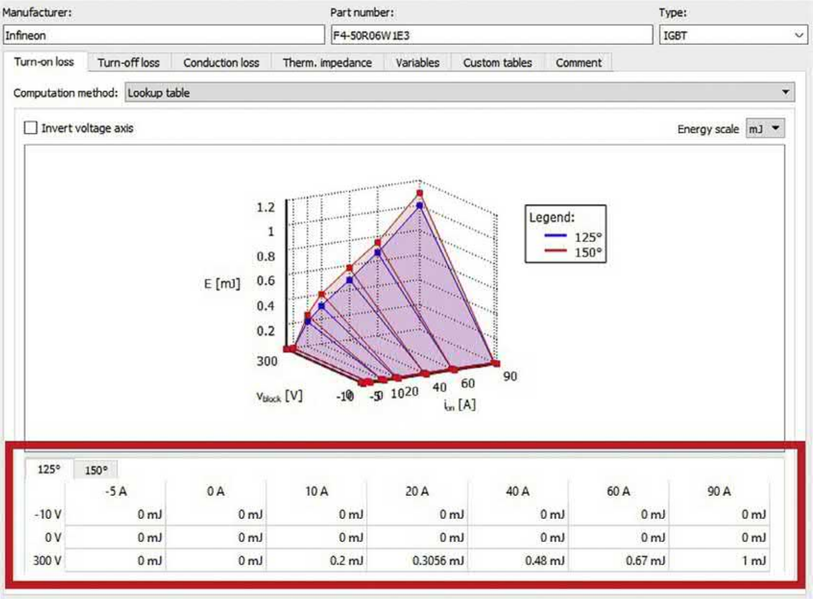
FIG. 5.32 Switching losses versus collector current. Eon shows the turn-on losses, while the Eoff shows the turn-off losses. *(Copyright with kind permission by Infineon Technologies AG, Germany.)*


Manufacturer: Infineon
Part number: F4-50R06W1E3
Type: IGBT
method: Lookup table
☐ Invert voltage axis
Energy scale mJ

Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment

FIG. 5.33 Turn-on tab is used to enter the turn-on losses data.

You can add a new voltage row to the table by right clicking on one of the voltage rows and selecting the New voltage values (Fig. 5.38).


Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
E [mJ]
1.2
1
0.8
0.6
0.4
0.2
300
V_{block} [V]
-10
-5
0
10
20
40
60
90
i_{on} [A]
Legend:
125°
150°
125°
150°

Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: IGBT

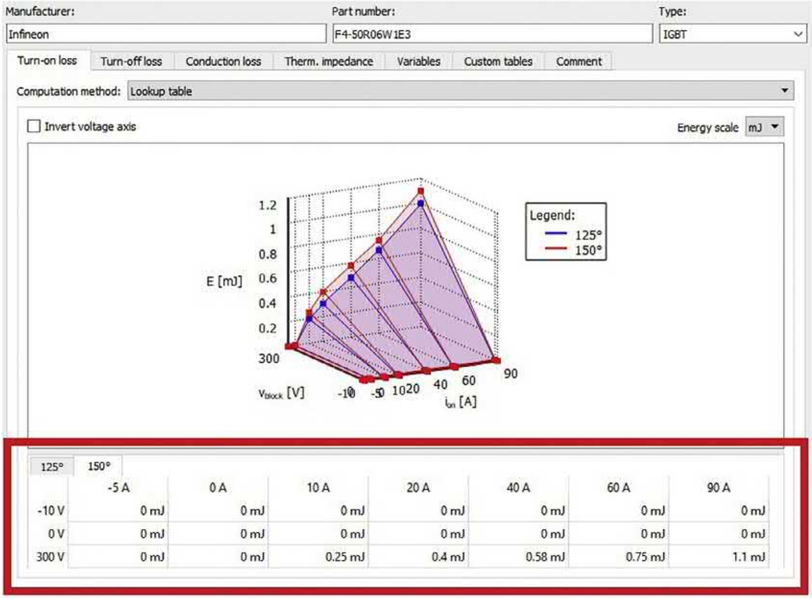
Computation method: Lookup table

☐ Invert voltage axis

Energy scale mJ

	-5 A	0 A	10 A	20 A	40 A	60 A	90 A
-10 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
0 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
300 V	0 mJ	0 mJ	0.2 mJ	0.3056 mJ	0.48 mJ	0.67 mJ	1 mJ

FIG. 5.34 Turn on switching losses for 125°C and different collector currents.


Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
E [mJ]
1.2
1
0.8
0.6
0.4
0.2
300
V_{block} [V]
-10
-5
0
10
20
40
60
90
i_{on} [A]
Legend:
125°
150°
125°
150°

Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: IGBT

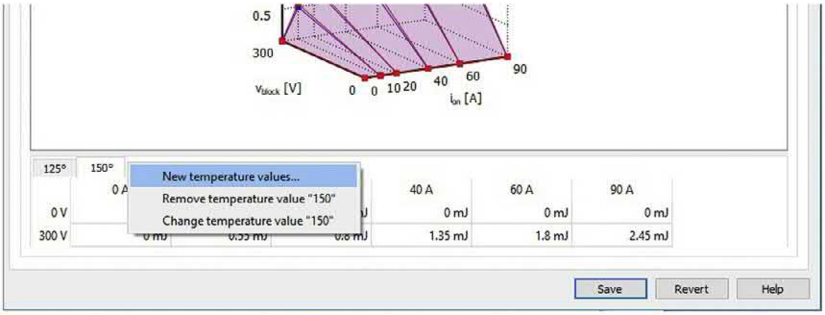
Computation method: Lookup table

☐ Invert voltage axis

Energy scale mJ

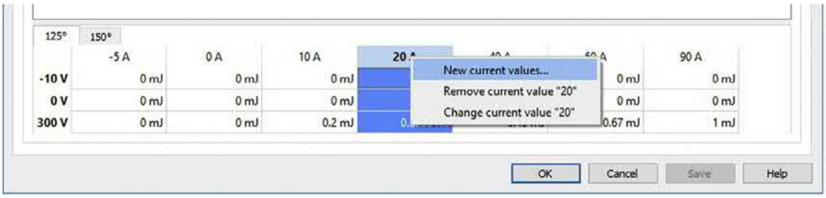
	-5 A	0 A	10 A	20 A	40 A	60 A	90 A
-10 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
0 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
300 V	0 mJ	0 mJ	0.25 mJ	0.4 mJ	0.58 mJ	0.75 mJ	1.1 mJ

FIG. 5.35 Turn on switching losses for 150°C and different collector currents.


0.5
300
V_{block} [V]
0
0
10
20
40
60
90
i_{on} [A]
125°
150°
New temperature values...
Remove temperature value "150"
Change temperature value "150"
Save
Revert
Help

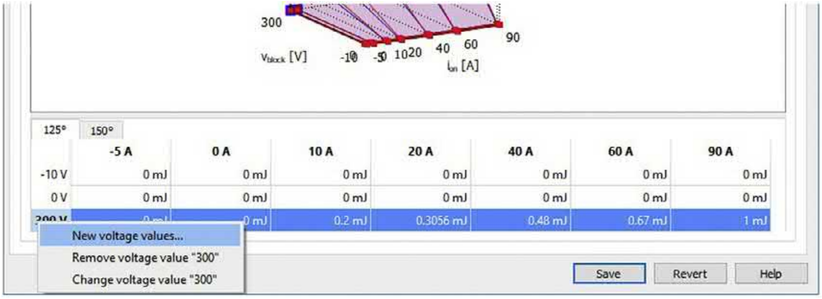
	0 A			40 A	60 A	90 A
0 V	[illegible]	[illegible]	[illegible]	0 mJ	0 mJ	0 mJ
300 V	0 mJ	0.55 mJ	0.8 mJ	1.35 mJ	1.8 mJ	2.45 mJ

FIG. 5.36 Addition of new temperature data points.


125°
150°
New current values...
Remove current value "20"
Change current value "20"
OK
Cancel
Save
Help

	-5 A	0 A	10 A	20 A	40 A	60 A	90 A
-10 V	0 mJ	0 mJ	0 mJ	[illegible]	[illegible]	0 mJ	0 mJ
0 V	0 mJ	0 mJ	0 mJ	[illegible]	[illegible]	0 mJ	0 mJ
300 V	0 mJ	0 mJ	0.2 mJ	0.[illegible]	[illegible]	0.67 mJ	1 mJ

FIG. 5.37 Addition of new current data point.


300
V_{block} [V]
-10
0
-5
0
10
20
40
60
90
i_{on} [A]
125°
150°
New voltage values...
Remove voltage value "300"
Change voltage value "300"
Save
Revert
Help

	-5 A	0 A	10 A	20 A	40 A	60 A	90 A
-10 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
0 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
300 V	0 mJ	0 mJ	0.2 mJ	0.3056 mJ	0.48 mJ	0.67 mJ	1 mJ

FIG. 5.38 Addition of new voltage data point.

5.5.2 Turn-off switching losses for the IGBT

Graph of switching loss of IGBT is redrawn in Fig. 5.39 for ease of reference. We use the E_{off} curves to model the turn-off switching losses of the IGBT.

In order to model the turn-off switching losses of IGBT, click the Turn-off loss tab (Fig. 5.40).

Enter the dissipated energy according to the graph shown in Fig. 5.39 (use the E_{off} curve). To add a linear relationship between the switching losses and

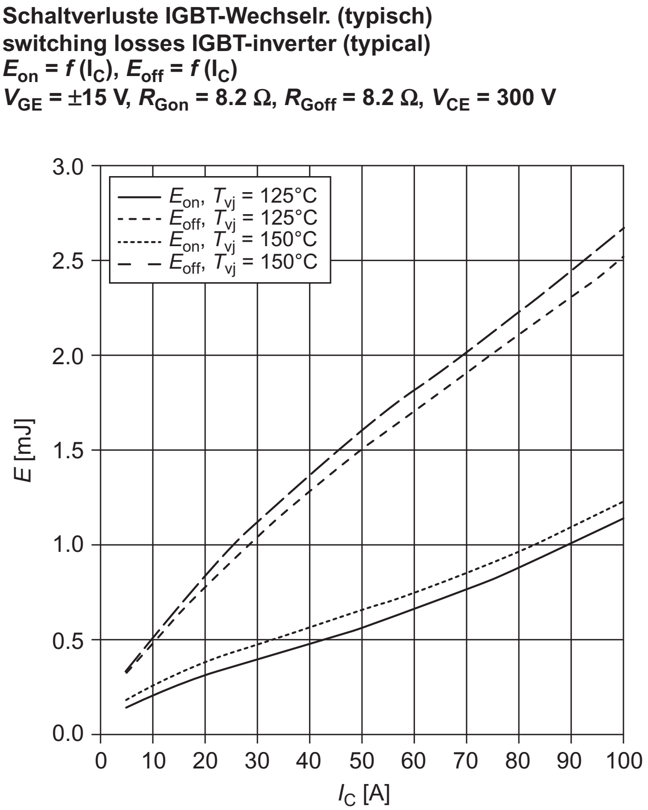
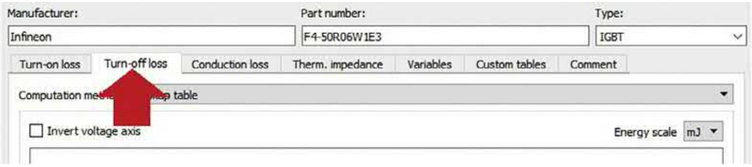


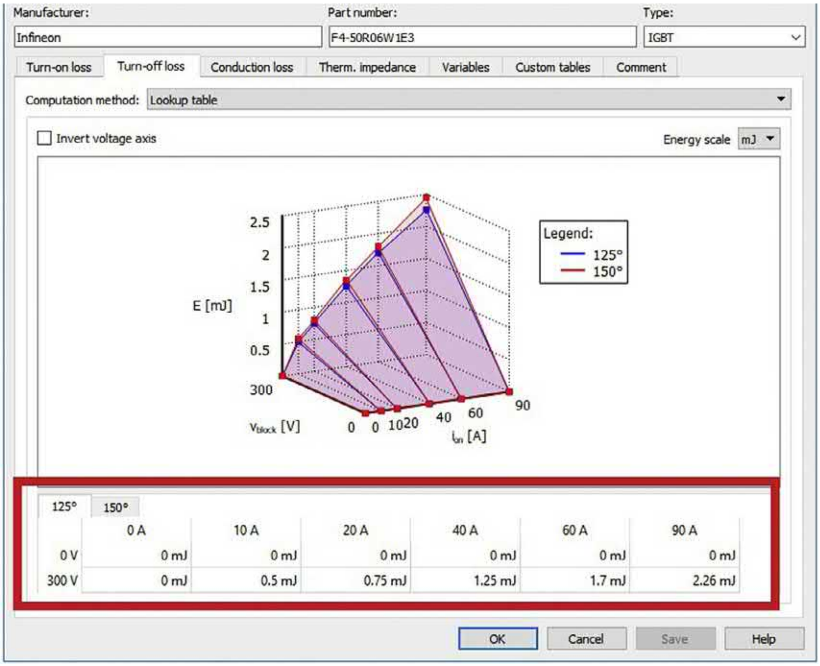
FIG. 5.39 Switching losses versus collector current. E_{off} shows the turn-off losses. *(Copyright with kind permission by Infineon Technologies AG, Germany.)*


Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment

Manufacturer: Infineon
Part number: F4-50R06W1E3
Type: IGBT
Computation me… …p table
☐ Invert voltage axis
Energy scale mJ

FIG. 5.40 Turn-off loss tab of thermal editor.

voltage, extend loss line to 0 V by entering loss values 0 mJ at 0 V for all currents (Figs. 5.41 and 5.42).



Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: IGBT

Computation method: Lookup table

☐ Invert voltage axis

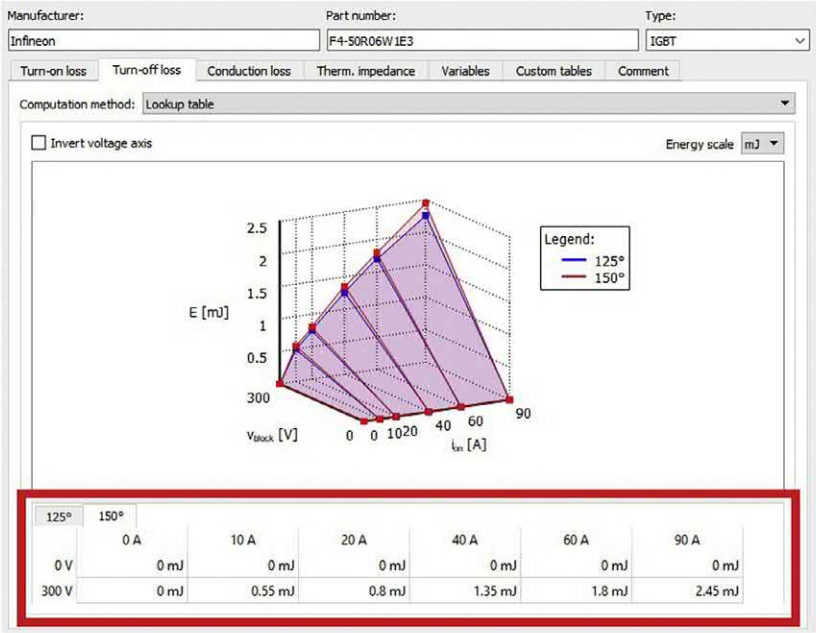
Energy scale: mJ

125°

	0 A	10 A	20 A	40 A	60 A	90 A
0 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
300 V	0 mJ	0.5 mJ	0.75 mJ	1.25 mJ	1.7 mJ	2.26 mJ

Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
E [mJ]
2.5
2
1.5
1
0.5
300
V_{block} [V]
0
0
10
20
40
60
90
i_{on} [A]
Legend:
125°
150°
125°
150°
OK
Cancel
Save
Help

FIG. 5.41 Turn-off switching losses for 125°C and different collector currents.



Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: IGBT

Computation method: Lookup table

☐ Invert voltage axis

Energy scale: mJ

150°

	0 A	10 A	20 A	40 A	60 A	90 A
0 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
300 V	0 mJ	0.55 mJ	0.8 mJ	1.35 mJ	1.8 mJ	2.45 mJ

Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
E [mJ]
2.5
2
1.5
1
0.5
300
V_{block} [V]
0
0
10
20
40
60
90
i_{on} [A]
Legend:
125°
150°
125°
150°

FIG. 5.42 Turn-on switching losses for 150°C and different collector currents.

5.6 Conduction losses for the IGBT

According to Fig. 5.25, during the transients some power is lost. The switching losses are not the only losses of a semiconductor switch. After transient is passed and the device reaches its steady state, some power are lost in form of heat, since the junction of switch has its own resistance and resistances dissipate energy in the form of heat according to $P = RI^2$ equation. Such a power loss is called conduction power loss.

Graph of collector current vs. collector-emitter voltage (for different junction temperatures) is shown in Fig. 5.43. We use this graph to model the conduction losses of the IGBT.

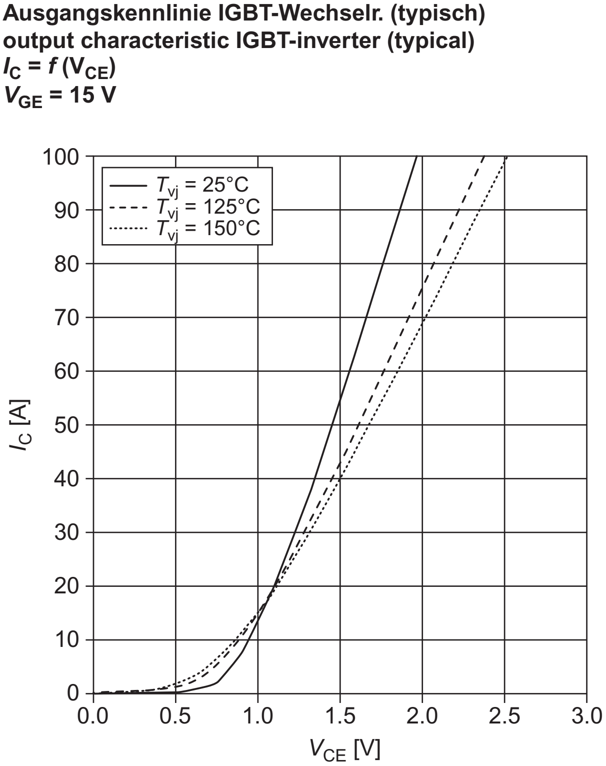
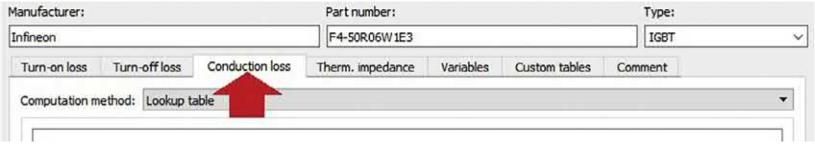


FIG. 5.43 Graph of collector current versus collector-emitter voltage for different temperatures. *(Copyright with kind permission by Infineon Technologies AG, Germany.)*

Click the Conduction loss tab in order to start modeling the conduction losses (Fig. 5.44).



Manufacturer: Infineon

Part number: F4-50R06W1E3

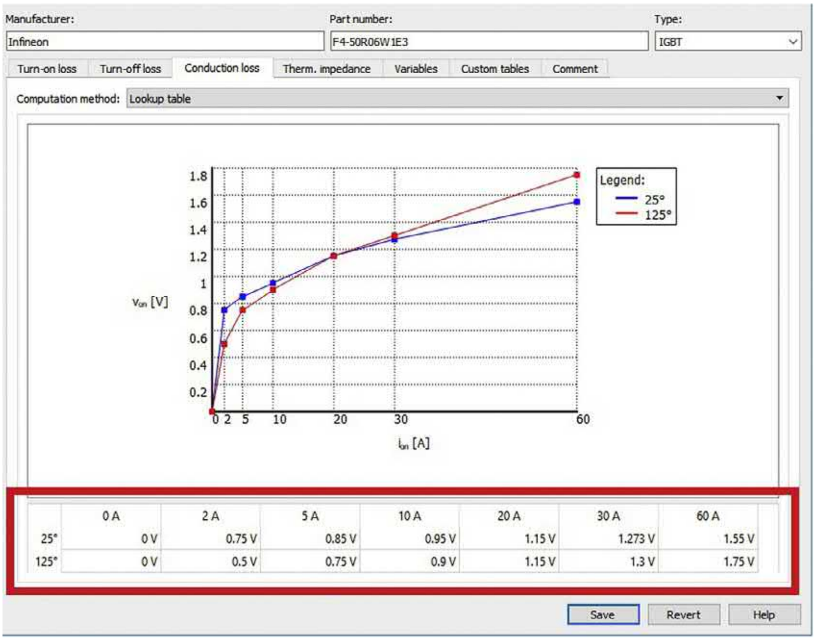
Type: IGBT

Computation method: Lookup table

Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment

FIG. 5.44 Conduction loss tab of thermal editor.

The data sheet provides the conduction losses for the IGBT for three different temperatures: 25°C, 125°C, and 150°C. Since the final junction temperature of the devices is not expected to exceed 125°C, it is sufficient to define the conduction losses at 25°C and 125°C (Fig. 5.45). You can enter the data for 150°C if you prefer to do so.



Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: IGBT

Computation method: Lookup table

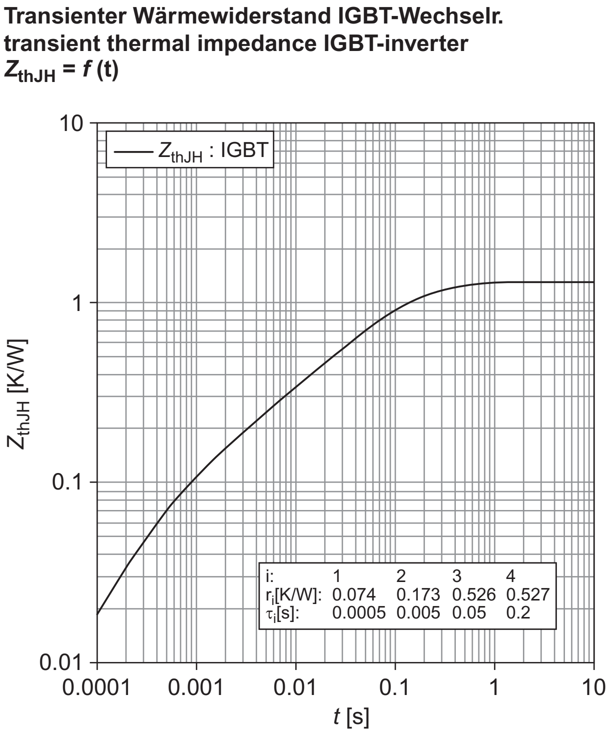
	0 A	2 A	5 A	10 A	20 A	30 A	60 A
25°	0 V	0.75 V	0.85 V	0.95 V	1.15 V	1.273 V	1.55 V
125°	0 V	0.5 V	0.75 V	0.9 V	1.15 V	1.3 V	1.75 V

Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
V_{on} [V]
1.8
1.6
1.4
1.2
1
0.8
0.6
0.4
0.2
0
2
5
10
20
30
60
i_{on} [A]
Legend:
25°
125°
Save
Revert
Help

FIG. 5.45 Collector-emitter voltage drop for different collector currents and junction temperatures.

5.7 Thermal impedances

The junction-case transient thermal impedance curves for the IGBT and diode are displayed in the data sheet along with the coefficients for a four-stage Foster network. The junction-case transient thermal impedance curve for the IGBT is shown in Fig. 5.46.



i:	1	2	3	4
r_i[K/W]:	0.074	0.173	0.526	0.527
τ_i[s]:	0.0005	0.005	0.05	0.2

FIG. 5.46 Thermal impedance graph of IGBT. Parameters of the fourth-order Foster network are given by datasheet. *(Copyright with kind permission by Infineon Technologies AG, Germany.)*

The Foster coefficients should not be used unless the case temperature of the device is held constant. Due to the series-connected capacitors in the Foster network, temperature variations at the case end of the chain are immediately propagated back to the junction end of the chain, rendering the transient response invalid.

If the case temperature is not constant, it is better to represent the junction-case thermal impedance using a first-order Cauer network as depicted in Fig. 5.47.

The resistance can be obtained directly from the thermal impedance curve. The transient thermal impedance curve corresponds in system theory to the step response and therefore contains the full thermal description of the system. The equation for the thermal resistance is:

$$Z_{th}(t) = \frac{T_{j1} - T_j(t)}{P_1},$$

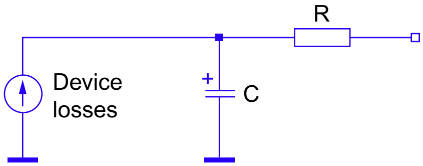
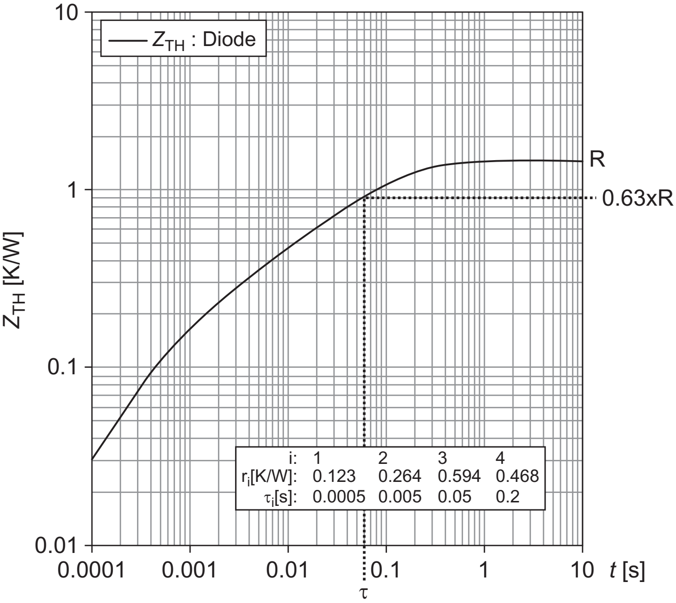


FIG. 5.47 First-order Cauer thermal impedance network.

where T_{j1} is a stationary temperature that the component is first heated to with a defined power dissipation P_1, and $T_{j1}(t)$ is the cooling temperature over time after the power dissipation is reduced to zero.

The transient thermal impedance curve for the diode is shown in Fig. 5.48. This curve is used to model the thermal impedance of the body diode.



i:	1	2	3	4
r_i[K/W]:	0.123	0.264	0.594	0.468
τ_i[s]:	0.0005	0.005	0.05	0.2

FIG. 5.48 The transient thermal impedance curve for the body diode. Parameters of the fourth-order Foster network are given by datasheet. *(Copyright with kind permission by Infineon Technologies AG, Germany.)*

The capacitance is calculated using:

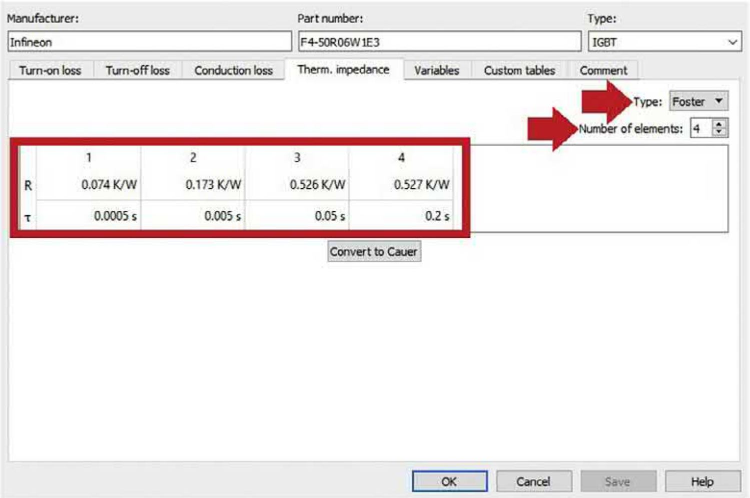
$$C = \frac{\tau}{R},$$

where τ, the time constant, is measured as the time taken for the resistance to reach 63% of its final value (see Fig. 5.48).

Note: The Foster representation of a thermal impedance is a mathematical representation of a thermal impedance that bears no resemblance to a physical structure. The internal node voltages do not represent the temperature distribution in a real system, and to extend the network to accommodate a heat sink, all Foster coefficients for the entire thermal network would need to be recalculated. Internally, PLECS always uses the Cauer network to calculate the thermal transitions. Foster networks are converted to Cauer networks at simulation start. Strictly speaking, this conversion is only accurate if the temperature at the outer end of the network, i.e., the case, is held constant.

For practical purposes, the conversion should yield accurate results if the external thermal capacitance is much bigger than the capacitances within the network.

In order to enter the thermal impedance of IGBT, click the Therm. Impedance tab (Fig. 5.49). Enter the four Foster parameters provided by the datasheet (Fig. 5.46).



Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: IGBT

Type: Foster

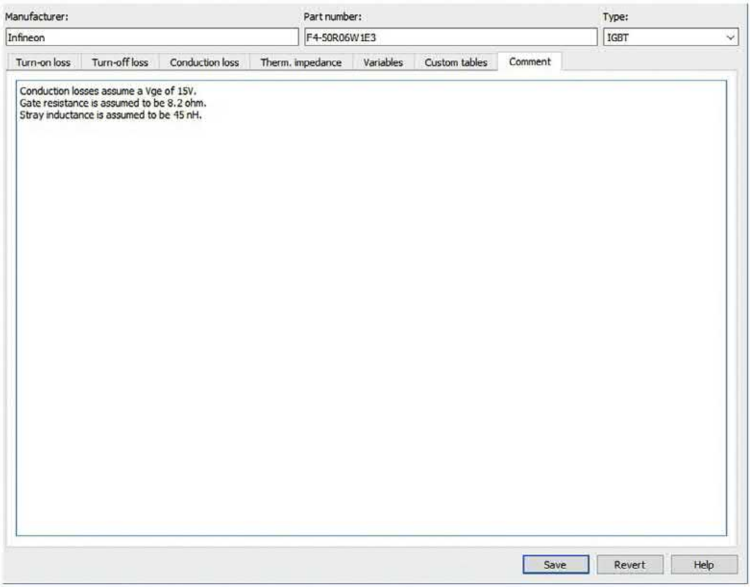
Number of elements: 4

	1	2	3	4
R	0.074 K/W	0.173 K/W	0.526 K/W	0.527 K/W
τ	0.0005 s	0.005 s	0.05 s	0.2 s

FIG. 5.49 Entering the thermal impedance of IGBT. The numbers are entered according to Fig. 5.46.

5.8 Adding comments

You can add required comments by using the Comment tab. Enter the desired comments in the provided text box (Fig. 5.50).


Manufacturer:
Infineon
Part number:
F4-50R06W1E3
Type:
IGBT
Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
Save
Revert
Help

Conduction losses assume a Vge of 15V.
Gate resistance is assumed to be 8.2 ohm.
Stray inductance is assumed to be 45 nH.

FIG. 5.50 The comment tab of thermal editor can be used when you need to add comment to the produced model.

5.9 Saving the produced thermal model

The produced model can be saved with the aid of File menu (Fig. 5.51).

After Save is clicked, a window will appear, which permits you to save the produced model in the desired path of your computer. PLECS saves the file with .xml extension (Fig. 5.52).

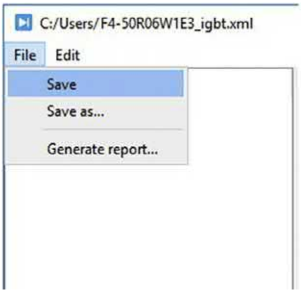
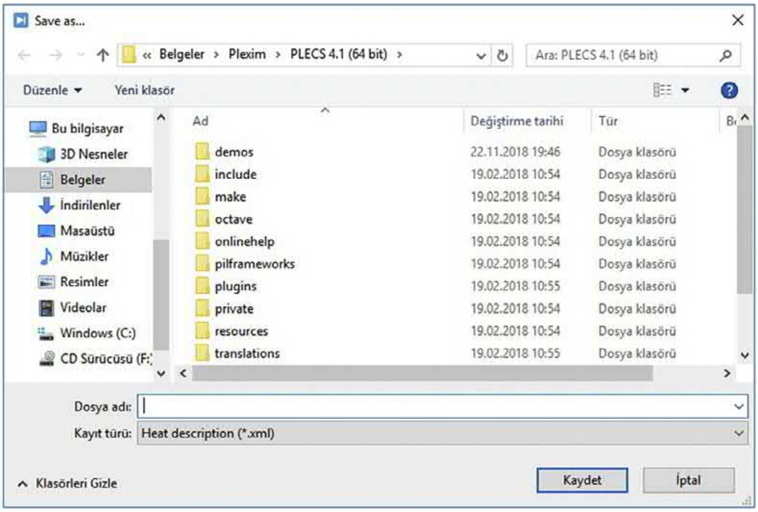

C:/Users/F4-50R06W1E3_igbt.xml
File
Edit
Save
Save as...
Generate report...

FIG. 5.51 Use the File menu to save the produced model. You can click the Save button (see Fig. 5.50) in order to save the produced model as well.


Save as...
« Belgeler › Plexim › PLECS 4.1 (64 bit) ›
Ara: PLECS 4.1 (64 bit)
Düzenle
Yeni klasör
Bu bilgisayar
3D Nesneler
Belgeler
İndirilenler
Masaüstü
Müzikler
Resimler
Videolar
Windows (C:)
CD Sürücüsü (F:)
Klasörleri Gizle
Kaydet
İptal

Ad	Değiştirme tarihi	Tür
demos	22.11.2018 19:46	Dosya klasörü
include	19.02.2018 10:54	Dosya klasörü
make	19.02.2018 10:54	Dosya klasörü
octave	19.02.2018 10:54	Dosya klasörü
onlinehelp	19.02.2018 10:54	Dosya klasörü
pilframeworks	19.02.2018 10:54	Dosya klasörü
plugins	19.02.2018 10:55	Dosya klasörü
private	19.02.2018 10:54	Dosya klasörü
resources	19.02.2018 10:54	Dosya klasörü
translations	19.02.2018 10:55	Dosya klasörü

Dosya adı:
Kayıt türü: Heat description (*.xml)

FIG. 5.52 Save as window.

5.10 Adding the produced model to thermal search path of PLECS

After you saved the model, you must add it to thermal search path of PLECS. This lets you to use the produced model in your simulations easily and quickly. Click the PLECS Preferences in order to add the produced model to thermal search path of PLECS (Fig. 5.53).

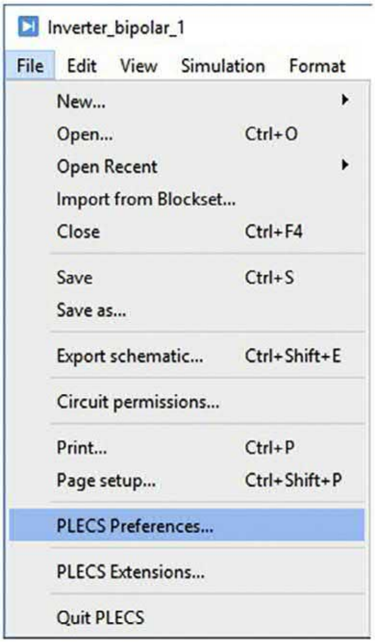
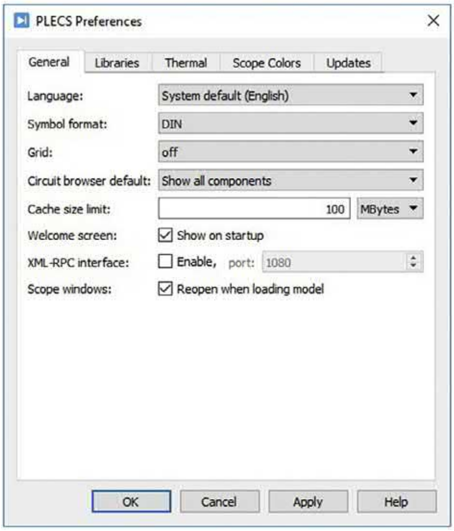

Inverter_bipolar_1
File
Edit
View
Simulation
Format
New...
Open... Ctrl+O
Open Recent
Import from Blockset...
Close Ctrl+F4
Save Ctrl+S
Save as...
Export schematic... Ctrl+Shift+E
Circuit permissions...
Print... Ctrl+P
Page setup... Ctrl+Shift+P
PLECS Preferences...
PLECS Extensions...
Quit PLECS

FIG. 5.53 PLECS Preferences can be found in the File menu.

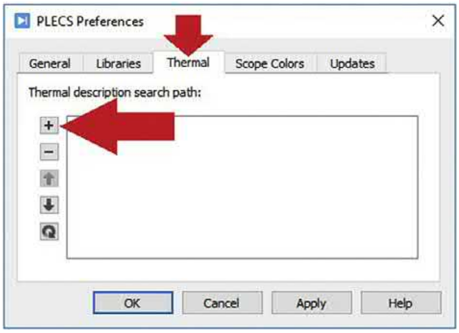
After you clicked the PLECS Preferences, the window shown in Fig. 5.54 will appear.


PLECS Preferences
General
Libraries
Thermal
Scope Colors
Updates
OK
Cancel
Apply
Help

Language: System default (English)
Symbol format: DIN
Grid: off
Circuit browser default: Show all components
Cache size limit: 100 MBytes
Welcome screen: ☑ Show on startup
XML-RPC interface: ☐ Enable, port: 1080
Scope windows: ☑ Reopen when loading model

FIG. 5.54 PLECS Preferences window.

Go to Thermal tab and then click the + button (Fig. 5.55).


PLECS Preferences
General
Libraries
Thermal
Scope Colors
Updates
+
−
OK
Cancel
Apply
Help

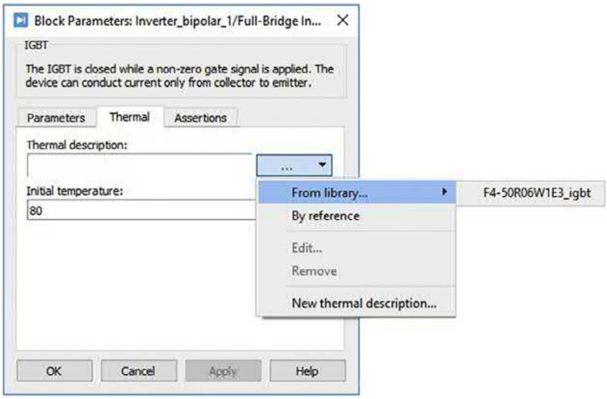
Thermal description search path:

FIG. 5.55 Thermal tab of PLECS Preferences window.

After you clicked the + button, PLECS opens the Select directory window. Go to the folder, which you saved the model and click the Select folder button in Select directory window. This adds the path of produced model to the thermal description search path of PLECS. There is no limitation on number of thermal descriptions in the thermal description search path. You can add as many as thermal models to the search path you want.

Once the model location is added to the thermal description search path of PLECS, you can add it to your simulations using the From library (Fig. 5.56).

Add the produced thermal model to IGBT1, IGBT2, IGBT3, and IGBT4 inside the subsystem model before proceeding.


Block Parameters: Inverter_bipolar_1/Full-Bridge In...

IGBT

The IGBT is closed while a non-zero gate signal is applied. The device can conduct current only from collector to emitter.

Thermal description:

Initial temperature: 80

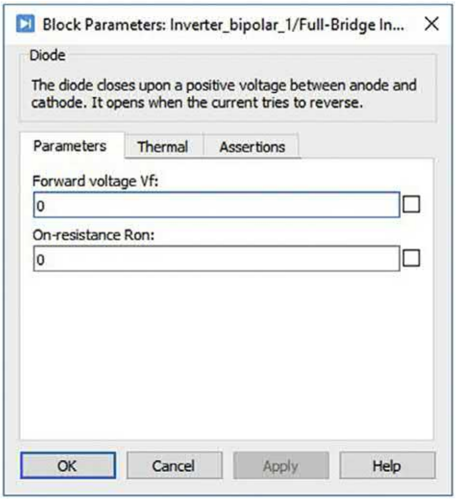
Parameters
Thermal
Assertions
...
From library...
F4-50R06W1E3_igbt
By reference
Edit...
Remove
New thermal description...
OK
Cancel
Apply
Help

FIG. 5.56 You can access the model after you add the produced model to Thermal description search path of PLECS.

5.11 Modeling losses of body diode

Up to now, we modeled the losses of IGBT. We must model the losses of body diode of IGBT as well. The process of modeling losses of diode is quite similar to the process of modeling losses of IGBT.

In order to model the losses of diodes, double click on one of the diodes inside the Full-bridge inverter subsystem. The window shown in Fig. 5.57 will appear.


Block Parameters: Inverter_bipolar_1/Full-Bridge In...

Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

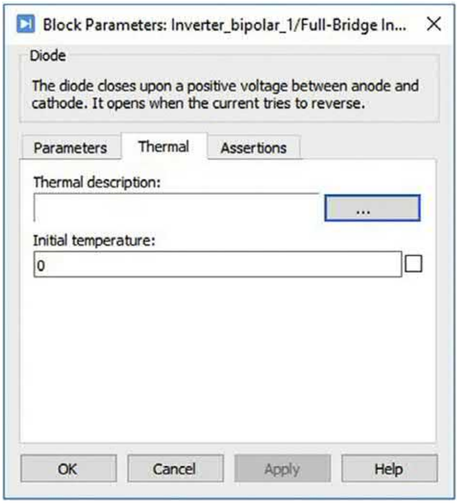
Forward voltage Vf: 0 ☐

On-resistance Ron: 0 ☐

Parameters
Thermal
Assertions
OK
Cancel
Apply
Help

FIG. 5.57 The diode settings.

Click the Thermal tab (Fig. 5.58).


Block Parameters: Inverter_bipolar_1/Full-Bridge In...

Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

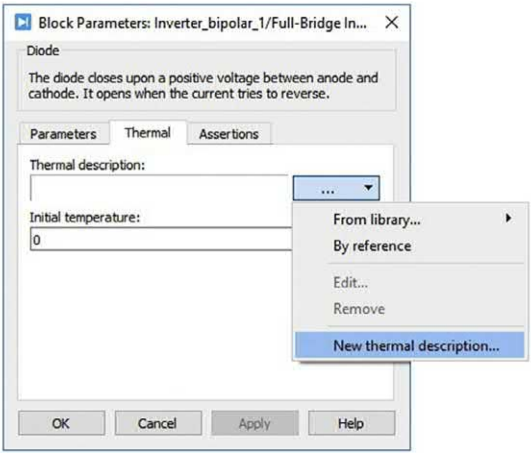
Thermal description:

Initial temperature: 0

Parameters
Thermal
Assertions
...
OK
Cancel
Apply
Help

FIG. 5.58 The Thermal tab of diode settings.

Click the Thermal description button and select the New thermal description (Fig. 5.59).


Block Parameters: Inverter_bipolar_1/Full-Bridge In...

Diode

The diode closes upon a positive voltage between anode and cathode. It opens when the current tries to reverse.

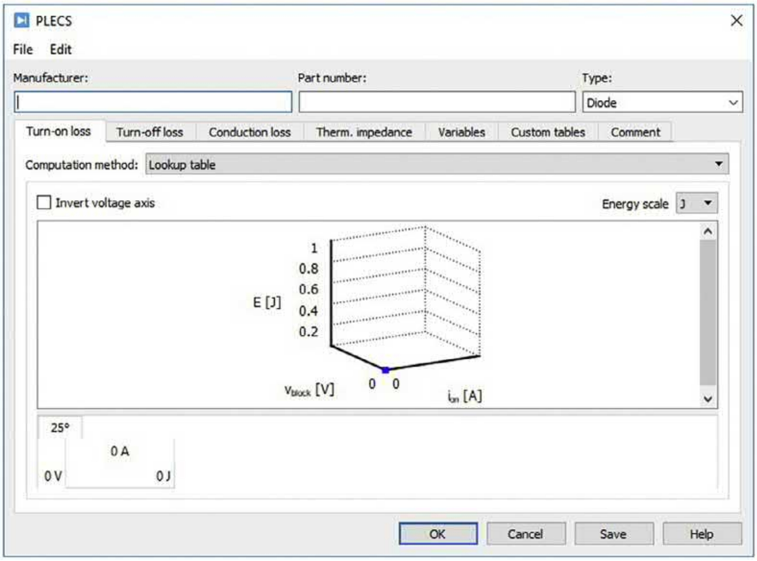
Thermal description:

Initial temperature: 0

Parameters
Thermal
Assertions
...
From library...
By reference
Edit...
Remove
New thermal description...
OK
Cancel
Apply
Help

FIG. 5.59 Use the New thermal description in order to define the diode losses.

The window shown in Fig. 5.60 will appear.


PLECS
File
Edit
Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
1
0.8
0.6
0.4
0.2
E [J]
V_{block} [V]
0
0
i_{on} [A]
25°
0 A
0 V
0 J
OK
Cancel
Save
Help

Manufacturer:

Part number:

Type: Diode

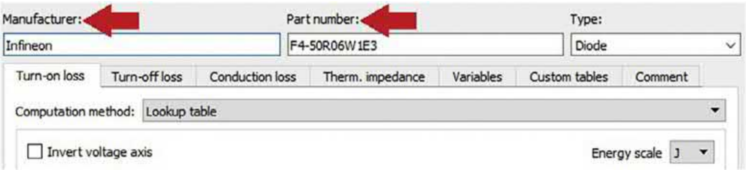
Computation method: Lookup table

☐ Invert voltage axis

Energy scale J

FIG. 5.60 The thermal editor window.

You can enter the manufacturer name and part number in the Manufacturer and Part number boxes, respectively (Fig. 5.61).


Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment

Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: Diode

Computation method: Lookup table

☐ Invert voltage axis

Energy scale J

FIG. 5.61 Entering the diode information.

5.11.1 The turn-on losses of body diode

Generally the turn-on losses of diode are negligible and can be ignored. The turn-off losses are important and must be modeled. So, you can ignore the turn-on loss and go to the Turn-off loss tab in order to start modeling the turn-off losses.

5.11.2 The turn-off losses of body diode

The turn-off losses curve for $V_{CE}=300\,\text{V}$ is shown in Fig. 5.62. The losses are given for two junction temperatures, namely, 125°C and 150°C.

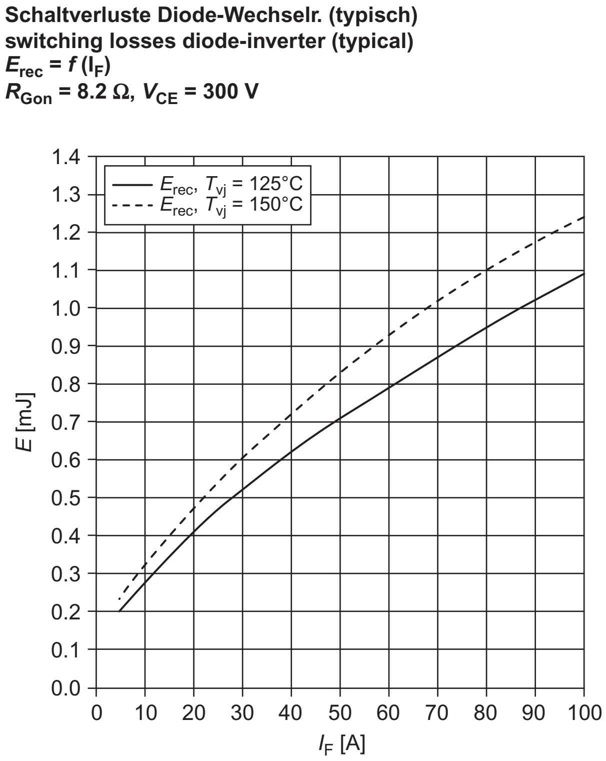


FIG. 5.62 Turn-off switching losses of body diode for different junction temperatures and currents. I_F shows the forward current of diode, i.e., the current from anode to cathode. The anode and cathode of body diode are connected to emitter and collector of IGBT, respectively. *(Copyright with kind permission by Infineon Technologies AG, Germany.)*

The turn-off tables are filled according to the curve shown in Fig. 5.62 (Figs. 5.63 and 5.64).

5.11.3 The conduction losses of body diode

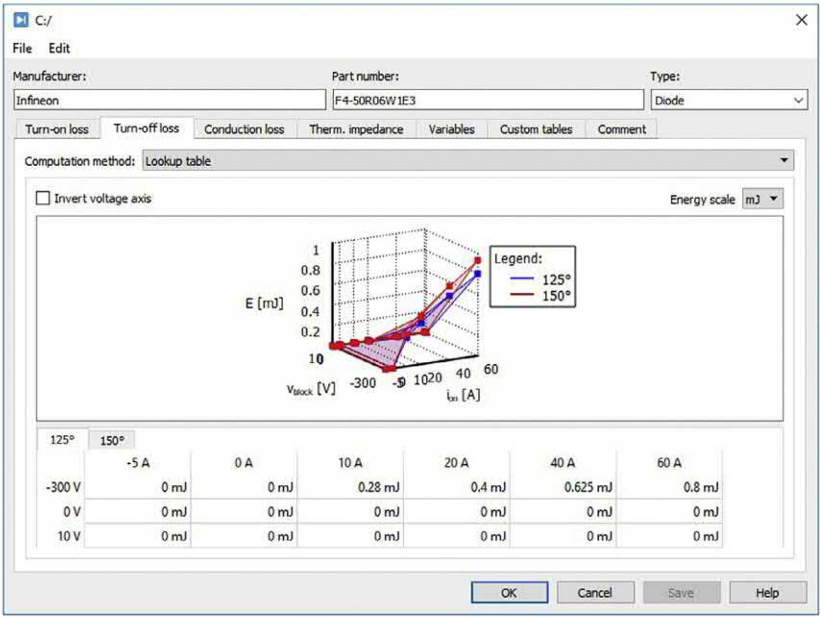
The curve for conduction losses of body diode is shown in Fig. 5.65.

The conduction losses tables are filled according to curve shown in Fig. 5.65. Since the final junction temperature of the devices is not expected to exceed 125°C, it is sufficient to define the conduction losses at 25°C and 125°C (Fig. 5.66). You can enter the data for 150°C if you prefer to do so.

5.12 Thermal impedance of body diode

The thermal impedance of body diode is shown in Fig. 5.67.

The thermal impedance table of body diode is filled according to Fig. 5.67.


C:/
File
Edit
Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
E [mJ]
1
0.8
0.6
0.4
0.2
10
V_block [V]
-300
-5
0
10
20
40
60
i_on [A]
Legend:
125°
150°
125°
150°
OK
Cancel
Save
Help

Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: Diode

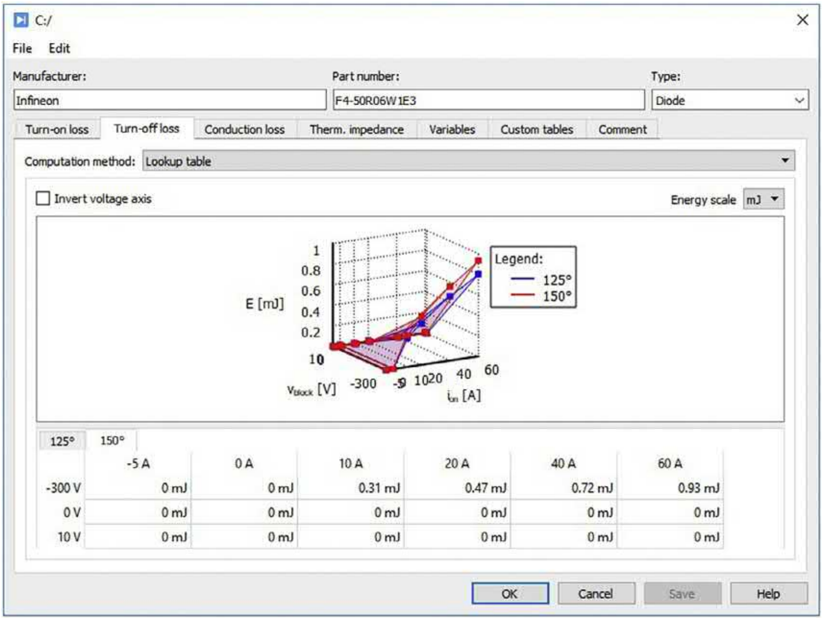
Computation method: Lookup table

☐ Invert voltage axis

Energy scale: mJ

	-5 A	0 A	10 A	20 A	40 A	60 A
-300 V	0 mJ	0 mJ	0.28 mJ	0.4 mJ	0.625 mJ	0.8 mJ
0 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
10 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ

FIG. 5.63 Turn-off switching losses for junction temperature of 125°C and different voltages ($V_{\text{anode}} - V_{\text{cathode}}$) and currents (current from anode to cathode).


C:/
File
Edit
Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
E [mJ]
1
0.8
0.6
0.4
0.2
10
V_block [V]
-300
-5
0
10
20
40
60
i_on [A]
Legend:
125°
150°
125°
150°
OK
Cancel
Save
Help

Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: Diode

Computation method: Lookup table

☐ Invert voltage axis

Energy scale: mJ

	-5 A	0 A	10 A	20 A	40 A	60 A
-300 V	0 mJ	0 mJ	0.31 mJ	0.47 mJ	0.72 mJ	0.93 mJ
0 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ
10 V	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ	0 mJ

FIG. 5.64 Turn-off switching losses for junction temperature of 150°C and different voltages ($V_{\text{anode}} - V_{\text{cathode}}$) and currents (current from anode to cathode).

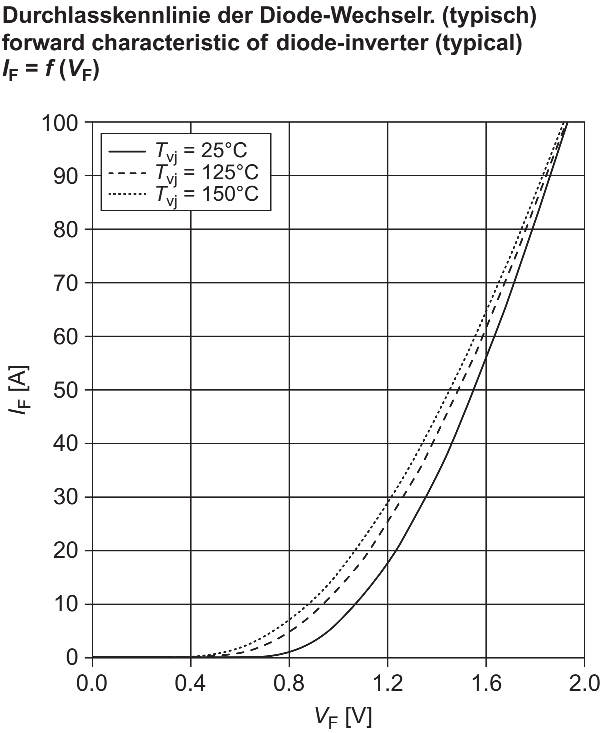
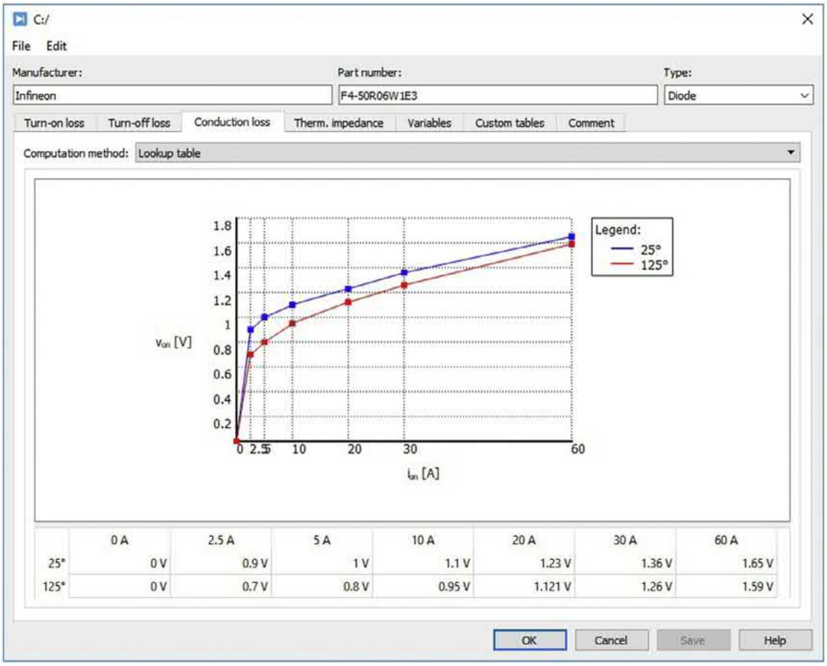


FIG. 5.65 Graph of diode forward current (current from anode to cathode) versus diode forward ($V_{anode} - V_{cathode}$) voltage drop. *(Copyright with kind permission by Infineon Technologies AG, Germany.)*


C:/
File
Edit
Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
1.8
1.6
1.4
1.2
1
0.8
0.6
0.4
0.2
V_{on} [V]
0
2.5
5
10
20
30
60
I_{on} [A]
Legend:
25°
125°
OK
Cancel
Save
Help

Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: Diode

Computation method: Lookup table

	0 A	2.5 A	5 A	10 A	20 A	30 A	60 A
25°	0 V	0.9 V	1 V	1.1 V	1.23 V	1.36 V	1.65 V
125°	0 V	0.7 V	0.8 V	0.95 V	1.121 V	1.26 V	1.59 V

FIG. 5.66 Entering the conduction loss of diode into the thermal editor.

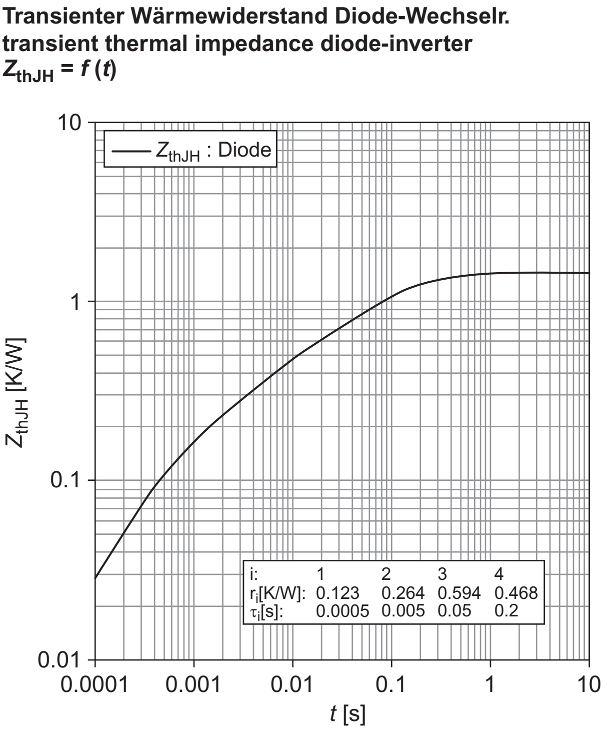
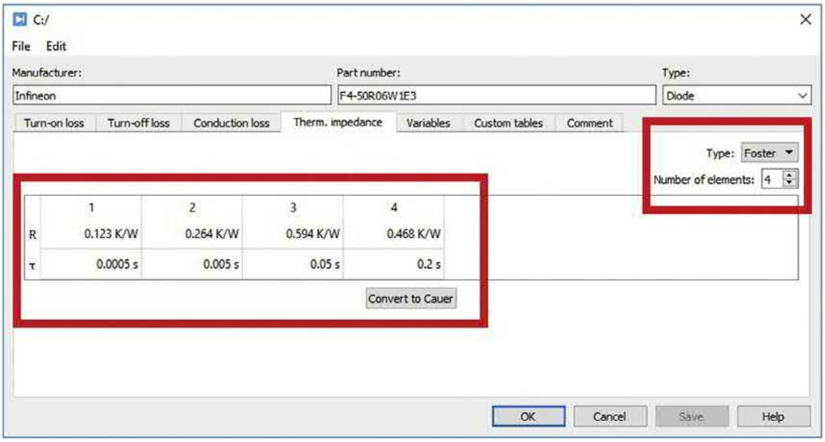


FIG. 5.67 Thermal impedance of body diode. Parameters of the fourth-order Foster network are given by datasheet. *(Copyright with kind permission by Infineon Technologies AG, Germany.)*

Modeling of body diode losses is finished here (Fig. 5.68). Save the produced model in the desired path as an .xml file, add the path to thermal search path, and add the produced model to all the diodes, i.e., D1, D2, D3, and D4, inside the Full-Bridge Inverter subsystem.


C:/
File Edit
Turn-on loss
Turn-off loss
Conduction loss
Therm. impedance
Variables
Custom tables
Comment
Convert to Cauer
OK
Cancel
Save
Help

Manufacturer: Infineon

Part number: F4-50R06W1E3

Type: Diode

Type: Foster

Number of elements: 4

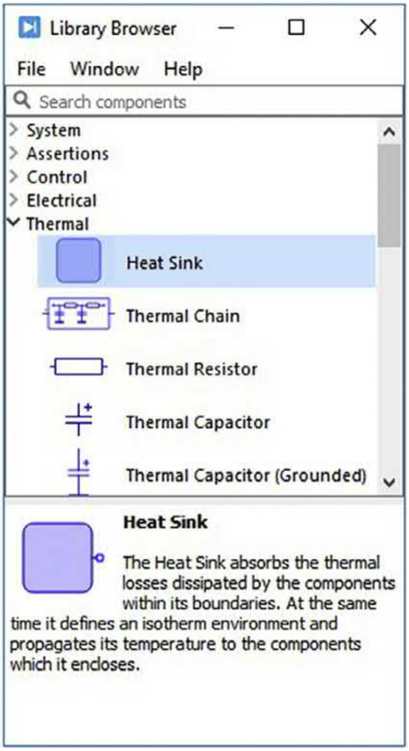
	1	2	3	4
R	0.123 K/W	0.264 K/W	0.594 K/W	0.468 K/W
τ	0.0005 s	0.005 s	0.05 s	0.2 s

FIG. 5.68 Entering the thermal impedance of body diode.

5.13 Loss measurements

We are ready to measure the inverter losses. As a starting point, assume a worst-case heat sink temperature of 80°C.

To set the heat sink temperature to this value, place a Heat Sink component (Fig. 5.69) over the Full-Bridge Inverter subsystem (Fig. 5.70). Set the heat sink thermal capacitance to 0 J/K (Fig. 5.71) and use a Constant Temperature (Grounded) thermal source component (Fig. 5.72) to set the heat sink temperature (Figs. 5.73 and 5.74). Also set the initial device temperatures to the same temperature as the initial heat sink temperature (by default the initial temperatures are 0°C).


Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Thermal
Heat Sink
Thermal Chain
Thermal Resistor
Thermal Capacitor
Thermal Capacitor (Grounded)

Heat Sink

The Heat Sink absorbs the thermal losses dissipated by the components within its boundaries. At the same time it defines an isotherm environment and propagates its temperature to the components which it encloses.

FIG. 5.69 The Heat Sink block can be found in the Thermal section of Library Browser.

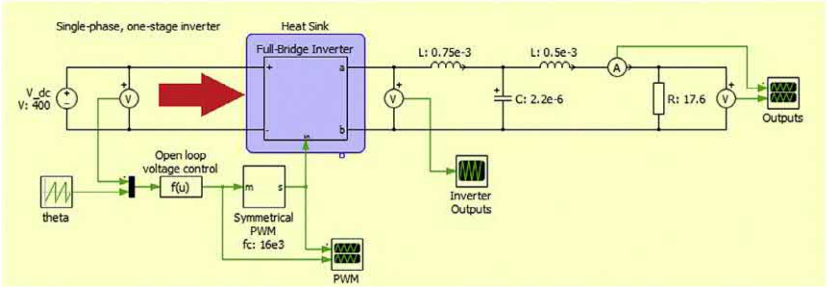
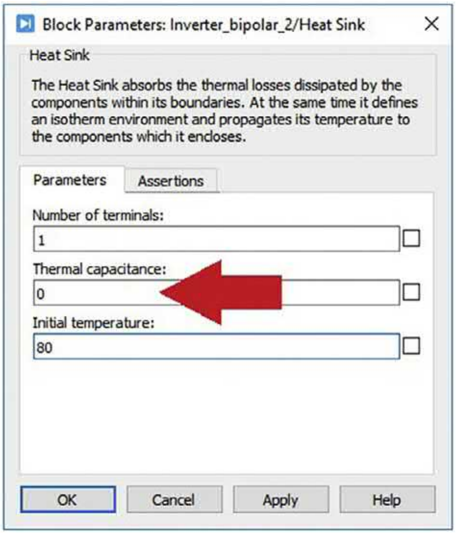

Single-phase, one-stage inverter
Heat Sink
Full-Bridge Inverter
V_dc
V: 400
L: 0.75e-3
L: 0.5e-3
C: 2.2e-6
R: 17.6
Outputs
Open loop
voltage control
f(u)
theta
Symmetrical
PWM
fc: 16e3
Inverter
Outputs
PWM

FIG. 5.70 Addition of Heat Sink block to the schematic.


Block Parameters: Inverter_bipolar_2/Heat Sink

Heat Sink

The Heat Sink absorbs the thermal losses dissipated by the components within its boundaries. At the same time it defines an isotherm environment and propagates its temperature to the components which it encloses.

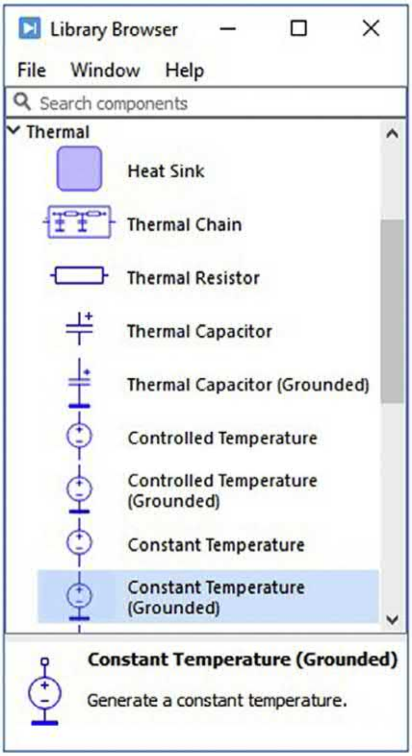
Number of terminals: 1

Thermal capacitance: 0

Initial temperature: 80

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 5.71 Double click on the heat sink block in order to set the thermal capacitance to zero. We are interested in the steady-state behavior, i.e., final values of temperatures, so the Thermal capacitance (which affects the transient behavior, not the steady-state behavior) is set to zero.


Library Browser
File
Window
Help
Search components
Thermal
Heat Sink
Thermal Chain
Thermal Resistor
Thermal Capacitor
Thermal Capacitor (Grounded)
Controlled Temperature
Controlled Temperature (Grounded)
Constant Temperature
Constant Temperature (Grounded)

Constant Temperature (Grounded)

Generate a constant temperature.

FIG. 5.72 The Constant Temperature (Grounded) block can be found in the Thermal section of Library Browser.

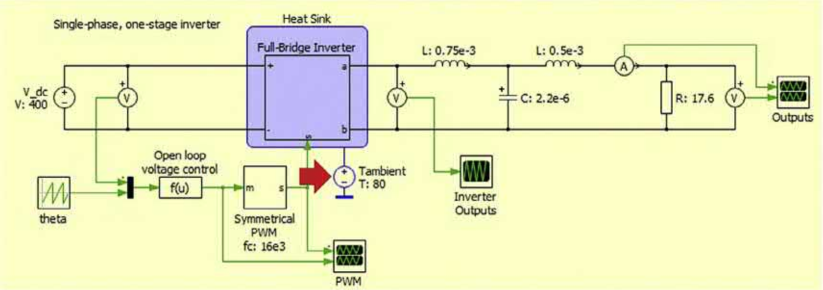
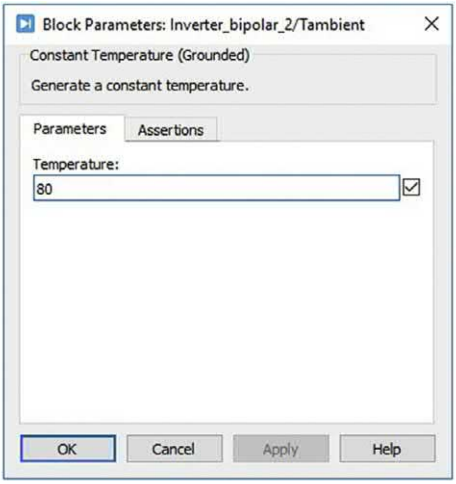

Single-phase, one-stage inverter
Heat Sink
Full-Bridge Inverter
V_dc
V: 400
L: 0.75e-3
L: 0.5e-3
C: 2.2e-6
R: 17.6
Outputs
Open loop
voltage control
f(u)
theta
Symmetrical
PWM
fc: 16e3
Tambient
T: 80
Inverter
Outputs
PWM

FIG. 5.73 The Constant Temperature (Grounded) is connected to the Heat Sink block to keep its temperature constant.


Block Parameters: Inverter_bipolar_2/Tambient

Constant Temperature (Grounded)

Generate a constant temperature.

Parameters
Assertions

Temperature: 80 ☑

OK
Cancel
Apply
Help

FIG. 5.74 Settings of Constant Temperature (Grounded) block.

The semiconductor losses are measured with the aid of loss calculation subsystems added to the schematic (Figs. 5.75 and 5.76). The inside of loss calculation subsystems is studied next.

5.13.1 Calculation of IGBT's losses

Here, we study the blocks, which calculate the IGBT1 losses. These blocks are shown in Fig. 5.77. The blocks, which calculate the losses for other IGBTs, are composed of same blocks. So, they are not studied.

The input of Loss Calculation IGBT1 is a probe block (Fig. 5.78). The Probe block can be found at System section of Library Browser.

Add a Probe block to the schematic (Fig. 5.79).

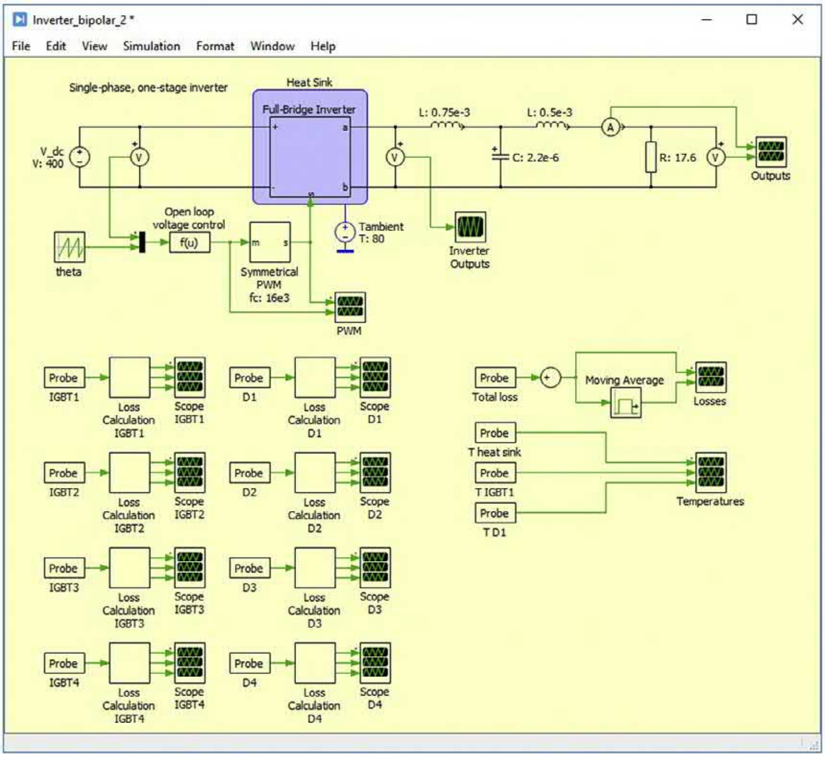

Inverter_bipolar_2 *
File Edit View Simulation Format Window Help
Single-phase, one-stage inverter
Heat Sink
Full-Bridge Inverter
a
b
V_dc
V: 400
V
L: 0.75e-3
L: 0.5e-3
A
C: 2.2e-6
R: 17.6
V
Outputs
theta
Open loop
voltage control
f(u)
m
s
Symmetrical
PWM
fc: 16e3
Tambient
T: 80
Inverter
Outputs
PWM
Probe
IGBT1
Loss
Calculation
IGBT1
Scope
IGBT1
Probe
D1
Loss
Calculation
D1
Scope
D1
Probe
IGBT2
Loss
Calculation
IGBT2
Scope
IGBT2
Probe
D2
Loss
Calculation
D2
Scope
D2
Probe
IGBT3
Loss
Calculation
IGBT3
Scope
IGBT3
Probe
D3
Loss
Calculation
D3
Scope
D3
Probe
IGBT4
Loss
Calculation
IGBT4
Scope
IGBT4
Probe
D4
Loss
Calculation
D4
Scope
D4
Probe
Total loss
Moving Average
Losses
Probe
T heat sink
Probe
T IGBT1
Probe
T D1
Temperatures

FIG. 5.75 Loss calculation and monitoring blocks are added to the schematic.

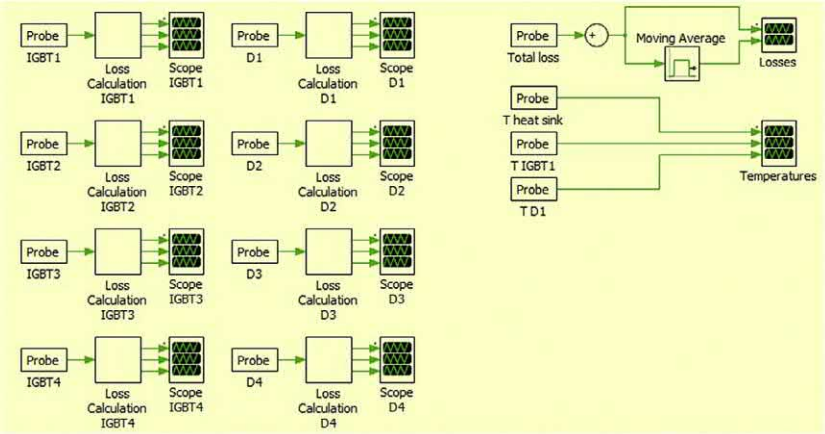

Probe
IGBT1
Loss
Calculation
IGBT1
Scope
IGBT1
Probe
D1
Loss
Calculation
D1
Scope
D1
Probe
Total loss
Moving Average
Losses
Probe
T heat sink
Probe
T IGBT1
Probe
T D1
Temperatures
Probe
IGBT2
Loss
Calculation
IGBT2
Scope
IGBT2
Probe
D2
Loss
Calculation
D2
Scope
D2
Probe
IGBT3
Loss
Calculation
IGBT3
Scope
IGBT3
Probe
D3
Loss
Calculation
D3
Scope
D3
Probe
IGBT4
Loss
Calculation
IGBT4
Scope
IGBT4
Probe
D4
Loss
Calculation
D4
Scope
D4

FIG. 5.76 Close up of loss calculation and monitoring blocks.

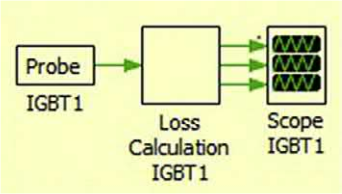
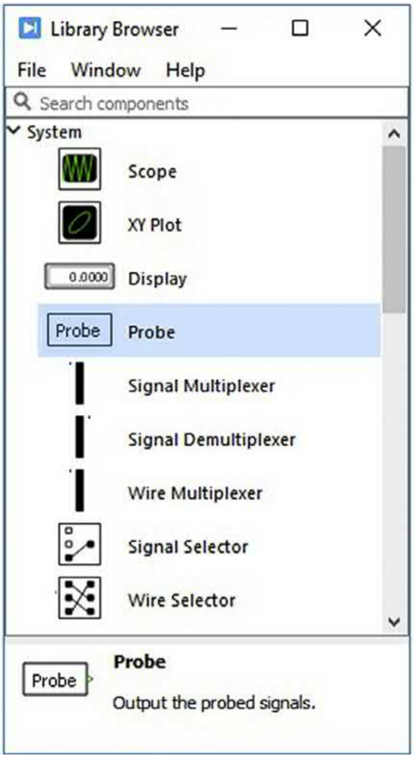

Probe
IGBT1
Loss
Calculation
IGBT1
Scope
IGBT1

FIG. 5.77 IGBT1 Probe block and Loss Calculation IGBT1 subsystem, measure the losses of IGBT1.


Library Browser
File
Window
Help
Search components
System
Scope
XY Plot
0.0000
Display
Probe
Probe
Signal Multiplexer
Signal Demultiplexer
Wire Multiplexer
Signal Selector
Wire Selector
Probe
Probe

Output the probed signals.

FIG. 5.78 The Probe block can be found in the System subsection of Library Browser.

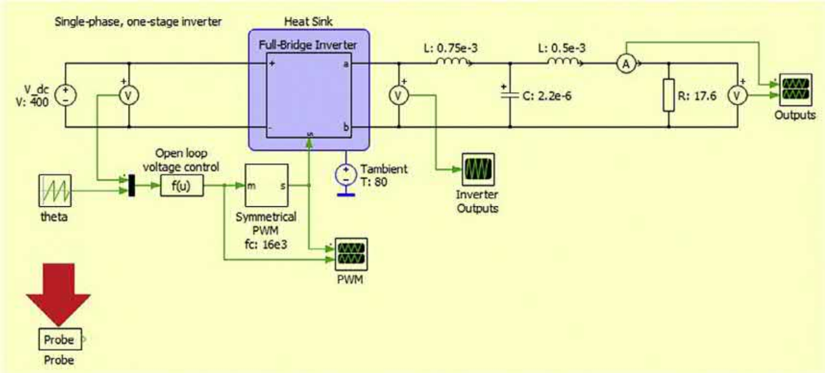

Single-phase, one-stage inverter
Heat Sink
Full-Bridge Inverter
+
a
-
b
L: 0.75e-3
L: 0.5e-3
A
V_dc
V: 400
V
V
+
C: 2.2e-6
R: 17.6
V
Outputs
Open loop
voltage control
f(u)
m
s
Symmetrical
PWM
fc: 16e3
Tambient
T: 80
Inverter
Outputs
theta
PWM
Probe
Probe

FIG. 5.79 A Probe block is added to the schematic.

Double click on the text below the Probe block to change it to IGBT1 (Fig. 5.80).

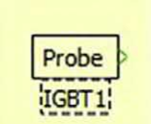

Probe
IGBT1

FIG. 5.80 Changing the Probe block label.

Open the Full-Bridge Inverter block. Drag and drop the IGBT1 inside the Full-Bridge Inverter block onto the probe block (Fig. 5.81).

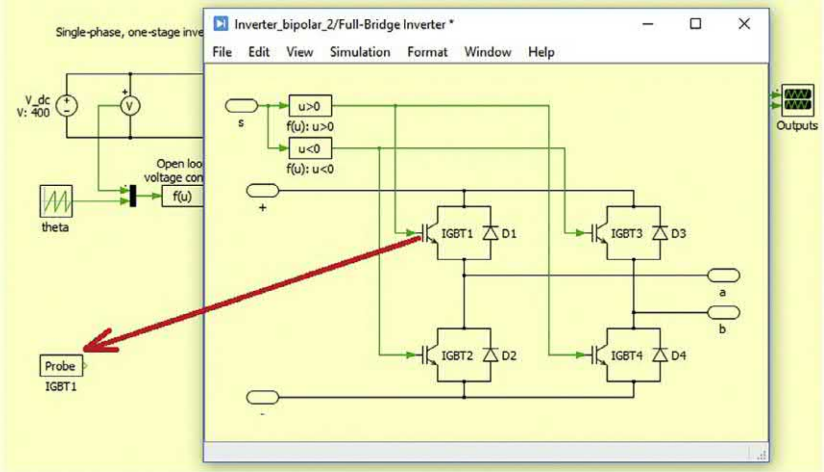
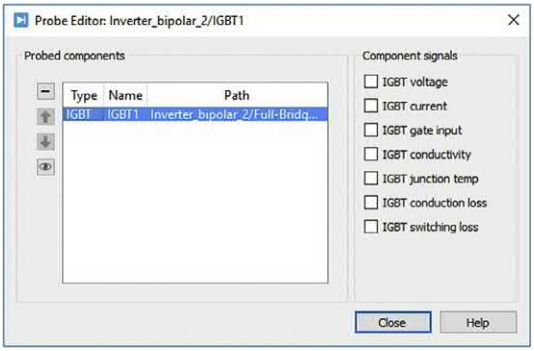

Single-phase, one-stage inve
V_dc
V: 400
V
Open loo
voltage con
f(u)
theta
Probe
IGBT1
Inverter_bipolar_2/Full-Bridge Inverter *
File
Edit
View
Simulation
Format
Window
Help
s
u>0
f(u): u>0
u<0
f(u): u<0
+
IGBT1
D1
IGBT3
D3
a
b
IGBT2
D2
IGBT4
D4
-
Outputs

FIG. 5.81 Drag and drop the IGBT1 from Full-Bridge Inverter subsystem onto the Probe block.

After you dropped the IGBT1 block onto the probe block, the window shown in Fig. 5.82 will appear.


Probe Editor: Inverter_bipolar_2/IGBT1

Probed components

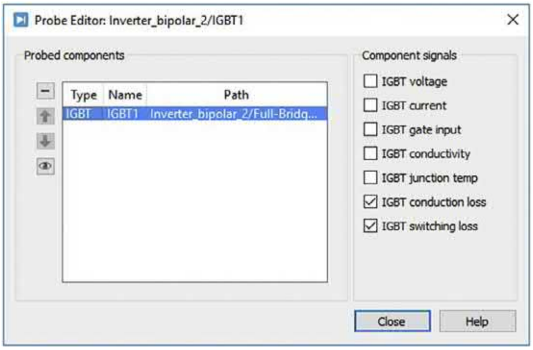
Type	Name	Path
IGBT	IGBT1	Inverter_bipolar_2/Full-Bridg...

Component signals

☐ IGBT voltage
☐ IGBT current
☐ IGBT gate input
☐ IGBT conductivity
☐ IGBT junction temp
☐ IGBT conduction loss
☐ IGBT switching loss

FIG. 5.82 The IGBT1 Probe block.

Select the IGBT conduction loss and IGBT switching loss (Fig. 5.83). So, the output of Probe block will be IGBT1 conduction loss and IGBT1 switching loss. The summation of these two will give us the total loss for IGBT1.


Probe Editor: Inverter_bipolar_2/IGBT1
Probed components
Component signals
Close
Help

Type	Name	Path
IGBT	IGBT1	Inverter_bipolar_2/Full-Bridg...

☐ IGBT voltage
☐ IGBT current
☐ IGBT gate input
☐ IGBT conductivity
☐ IGBT junction temp
☑ IGBT conduction loss
☑ IGBT switching loss

FIG. 5.83 Select the IGBT conduction loss and IGBT switching loss.

The inside of Loss Calculation IGBT1 subsystem is shown in Fig. 5.84. Two average blocks are used in order to calculate the average values of losses.

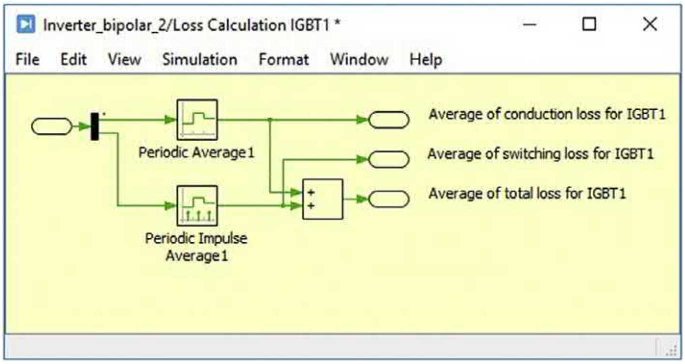
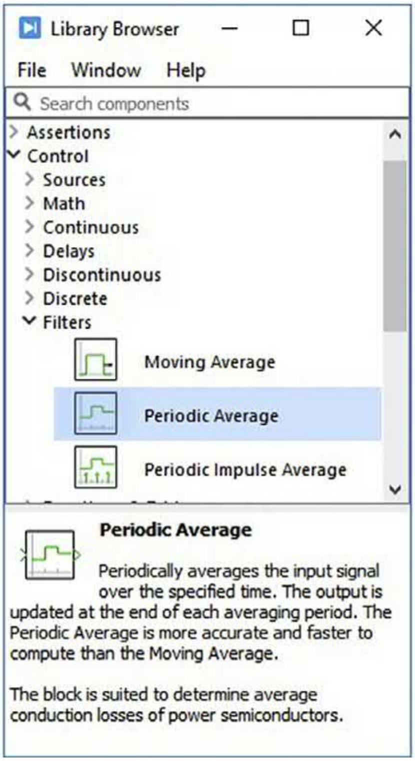

Inverter_bipolar_2/Loss Calculation IGBT1 *
File
Edit
View
Simulation
Format
Window
Help
Periodic Average1
Periodic Impulse Average1
+
+
Average of conduction loss for IGBT1
Average of switching loss for IGBT1
Average of total loss for IGBT1

FIG. 5.84 Calculation of average conduction loss, average switching loss, and total average loss (conduction loss + switching loss) for IGBT1.

The switching loss has an impulsive nature: It happens only when the switch status changes, i.e., when the on switch turns off or when the off switch turns on. So, we need to use the Periodic Impulse Average block in order to calculate the average of such a waveform. The conduction loss happens during the on period and its nature is not impulsive like the switching loss. So, average of conduction loss is calculated with a Periodic Average block.

The Periodic Average and Periodic Impulse Average can be found in the Filters section of Library Browser (Figs. 5.85 and 5.86).


Library Browser
File
Window
Help
Search components
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Moving Average
Periodic Average
Periodic Impulse Average

Periodic Average

Periodically averages the input signal over the specified time. The output is updated at the end of each averaging period. The Periodic Average is more accurate and faster to compute than the Moving Average.

The block is suited to determine average conduction losses of power semiconductors.

FIG. 5.85 The Periodic Average block can be found in the Filters section of Library Browser.

The switching frequency is 16 kHz. So, the averaging time of Periodic Average and Periodic Impulse Average blocks must be set to $\frac{1}{16000} = 62.5\ \mu\text{s}$. Settings of Periodic Average and Periodic Impulse Average blocks are shown in Figs. 5.87 and 5.88, respectively.

5.13.2 Calculation of body diode's losses

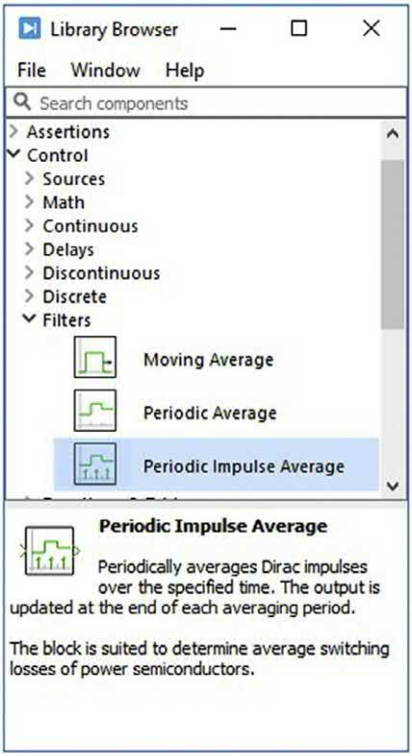
The losses for diodes are calculated in the same way. Here, we study the calculation of losses for diode D1only. The calculation of losses for D2, D3, and D4 is the same. Fig. 5.89 shows the blocks, which calculate the losses of D1.

The Probe block settings are shown in Fig. 5.90.

The inside of Loss Calculation D1 subsystem is shown in Fig. 5.91. It is the same as Fig. 5.84.

5.13.3 Calculation of total losses

The losses of IGBT1, IGBT2, IGBT3, IGBT4, D1, D2, D3, and D4 are calculated individually. In order to obtain the total losses, the losses of four IGBTs

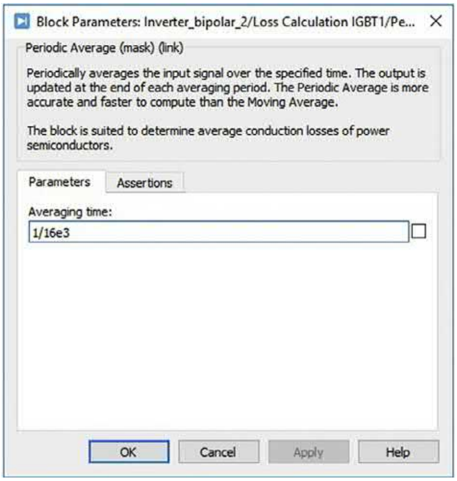

Library Browser
File
Window
Help
Search components
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Moving Average
Periodic Average
Periodic Impulse Average

Periodic Impulse Average

Periodically averages Dirac impulses over the specified time. The output is updated at the end of each averaging period.

The block is suited to determine average switching losses of power semiconductors.

FIG. 5.86 The Periodic Impulse Average block can be found in the Filters section of Library Browser.


Block Parameters: Inverter_bipolar_2/Loss Calculation IGBT1/Pe...

Periodic Average (mask) (link)

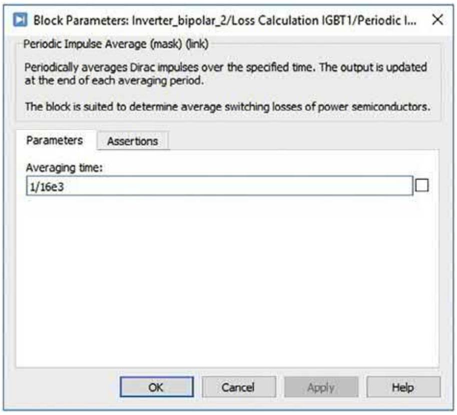
Periodically averages the input signal over the specified time. The output is updated at the end of each averaging period. The Periodic Average is more accurate and faster to compute than the Moving Average.

The block is suited to determine average conduction losses of power semiconductors.

Averaging time: 1/16e3 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 5.87 The Periodic Average block settings.


Block Parameters: Inverter_bipolar_2/Loss Calculation IGBT1/Periodic I...

Periodic Impulse Average (mask) (link)

Periodically averages Dirac impulses over the specified time. The output is updated at the end of each averaging period.

The block is suited to determine average switching losses of power semiconductors.

Averaging time: 1/16e3

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 5.88 The Periodic Impulse Average block settings.

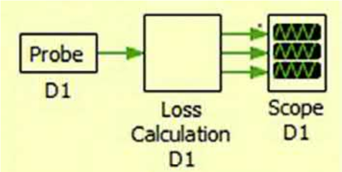
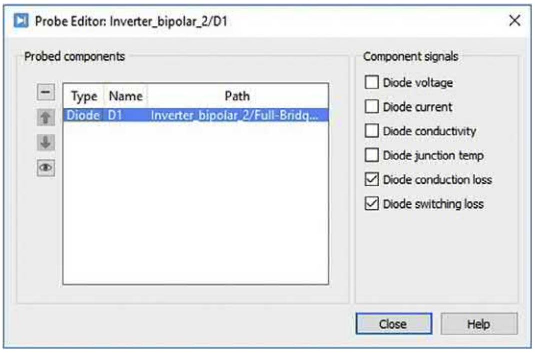

Probe
D1
Loss
Calculation
D1
Scope
D1

FIG. 5.89 D1 Probe block and Loss Calculation D1 subsystem measure the losses of diode D1.


Probe Editor: Inverter_bipolar_2/D1

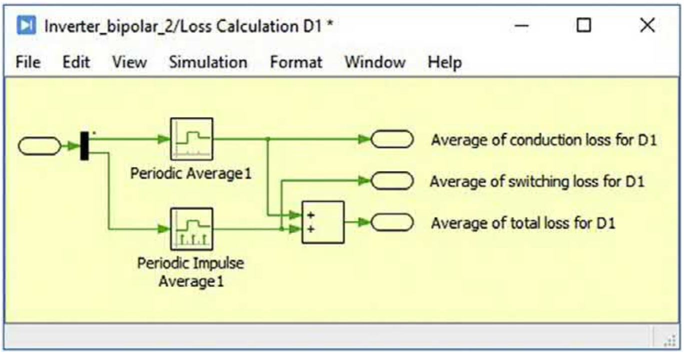
Probed components

Type	Name	Path
Diode	D1	Inverter_bipolar_2/Full-Bridg...

Component signals

☐ Diode voltage
☐ Diode current
☐ Diode conductivity
☐ Diode junction temp
☑ Diode conduction loss
☑ Diode switching loss

FIG. 5.90 The D1 Probe block.


Inverter_bipolar_2/Loss Calculation D1 *

File
Edit
View
Simulation
Format
Window
Help
Periodic Average1
Periodic Impulse
Average1
+
+
Average of conduction loss for D1
Average of switching loss for D1
Average of total loss for D1

FIG. 5.91 Calculation of average conduction loss, average switching loss, and total average loss (conduction loss+switching loss) for diode D1.

and for body diodes must be summed together. This is done with the aid of blocks shown in Fig. 5.92.

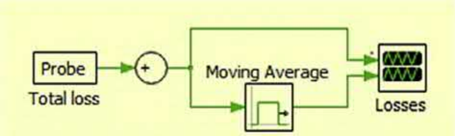

Probe
Total loss
+
Moving Average
Losses

FIG. 5.92 Calculation of overall total losses. The scope will show both the instantaneous and average losses.

In order to calculate the total losses, add a Probe block to the simulation and rename it to Total loss (Fig. 5.93).

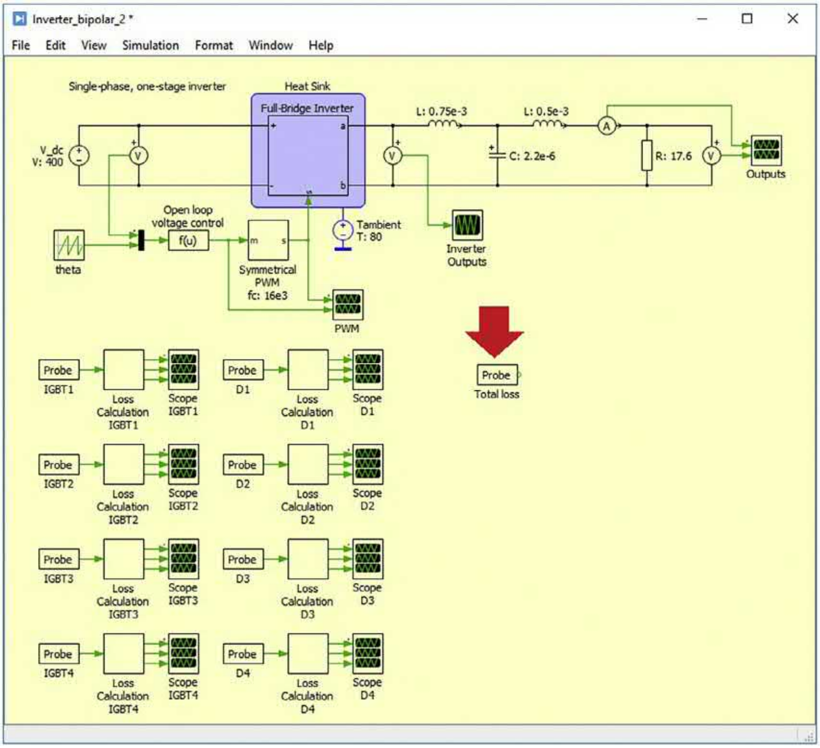
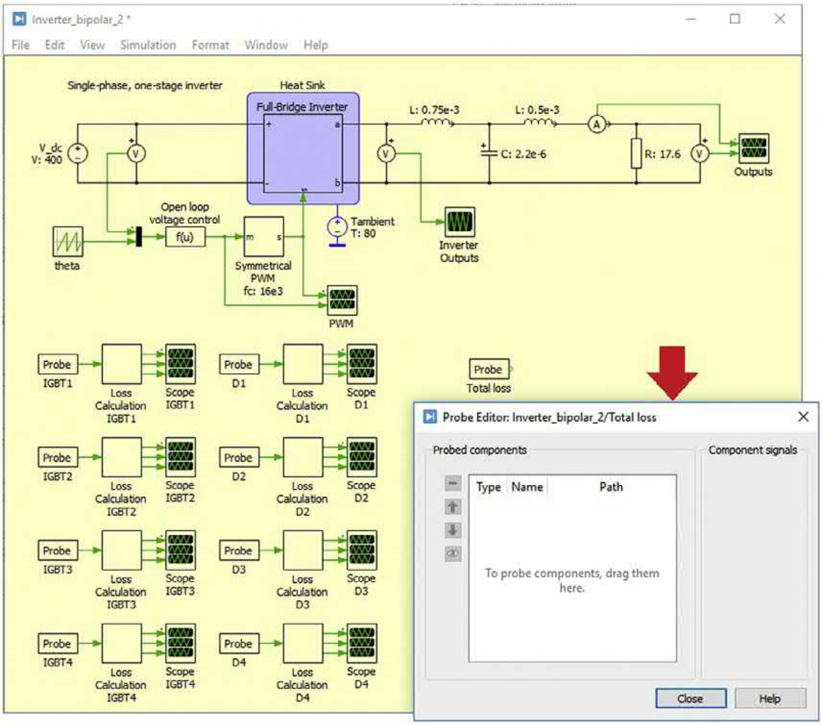

Inverter_bipolar_2 *
File
Edit
View
Simulation
Format
Window
Help
Single-phase, one-stage inverter
Heat Sink
Full-Bridge Inverter
L: 0.75e-3
L: 0.5e-3
V_dc
V: 400
C: 2.2e-6
R: 17.6
Outputs
Tambient
T: 80
Open loop
voltage control
f(u)
theta
Symmetrical
PWM
fc: 16e3
Inverter
Outputs
PWM
Probe
Total loss
Probe
IGBT1
Loss
Calculation
IGBT1
Scope
IGBT1
Probe
D1
Loss
Calculation
D1
Scope
D1
Probe
IGBT2
Loss
Calculation
IGBT2
Scope
IGBT2
Probe
D2
Loss
Calculation
D2
Scope
D2
Probe
IGBT3
Loss
Calculation
IGBT3
Scope
IGBT3
Probe
D3
Loss
Calculation
D3
Scope
D3
Probe
IGBT4
Loss
Calculation
IGBT4
Scope
IGBT4
Probe
D4
Loss
Calculation
D4
Scope
D4

FIG. 5.93 A probe named Total loss is added to the schematic.

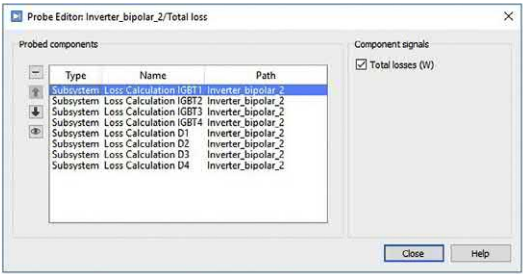
Double click the Total loss Probe block in order to open it (Fig. 5.94).


Inverter_bipolar_2 *
File Edit View Simulation Format Window Help
Single-phase, one-stage inverter
Heat Sink
Full-Bridge Inverter
L: 0.75e-3
L: 0.5e-3
V_dc
V: 400
C: 2.2e-6
R: 17.6
Outputs
Open loop
voltage control
f(u)
theta
Symmetrical
PWM
fc: 16e3
Tambient
T: 80
Inverter
Outputs
PWM
Probe
IGBT1
Loss
Calculation
IGBT1
Scope
IGBT1
Probe
D1
Loss
Calculation
D1
Scope
D1
Probe
Total loss
Probe
IGBT2
Loss
Calculation
IGBT2
Scope
IGBT2
Probe
D2
Loss
Calculation
D2
Scope
D2
Probe
IGBT3
Loss
Calculation
IGBT3
Scope
IGBT3
Probe
D3
Loss
Calculation
D3
Scope
D3
Probe
IGBT4
Loss
Calculation
IGBT4
Scope
IGBT4
Probe
D4
Loss
Calculation
D4
Scope
D4
Probe Editor: Inverter_bipolar_2/Total loss
Probed components
Component signals
Type Name Path
Close
Help

To probe components, drag them here.

FIG. 5.94 Total loss Probe block is opened.

Drag the Loss Calculation IGBT1, Loss Calculation IGBT2, Loss Calculation IGBT3, Loss Calculation IGBT4, Loss Calculation D1, Loss Calculation D2, Loss Calculation D3, and Loss Calculation D4 into the Total loss Probe block. Fig. 5.95 shows the Total loss Probe block after the drag and drops are finished.

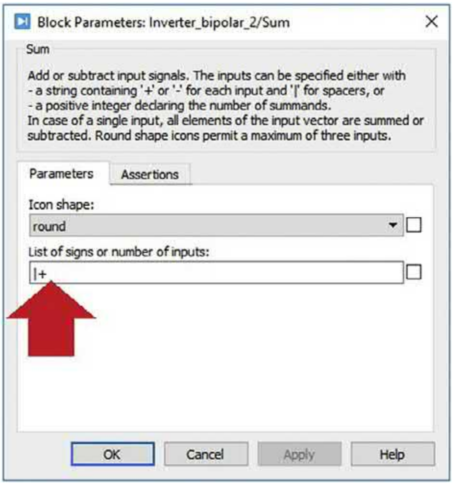

Probe Editor: Inverter_bipolar_2/Total loss
Probed components
Component signals
Close
Help

Type	Name	Path
Subsystem	Loss Calculation IGBT1	Inverter_bipolar_2
Subsystem	Loss Calculation IGBT2	Inverter_bipolar_2
Subsystem	Loss Calculation IGBT3	Inverter_bipolar_2
Subsystem	Loss Calculation IGBT4	Inverter_bipolar_2
Subsystem	Loss Calculation D1	Inverter_bipolar_2
Subsystem	Loss Calculation D2	Inverter_bipolar_2
Subsystem	Loss Calculation D3	Inverter_bipolar_2
Subsystem	Loss Calculation D4	Inverter_bipolar_2

☑ Total losses (W)

FIG. 5.95 All the subsystems are drag and dropped onto the Total loss Probe block.

Connect the output of the Total loss Probe block to a summation block (Fig. 5.92). The settings of sum block are shown in Fig. 5.96. With these settings, the sum output will be the summation of all the 8 inputs.


Block Parameters: Inverter_bipolar_2/Sum
Sum
Parameters
Assertions
OK
Cancel
Apply
Help

Add or subtract input signals. The inputs can be specified either with
- a string containing '+' or '-' for each input and '|' for spacers, or
- a positive integer declaring the number of summands.

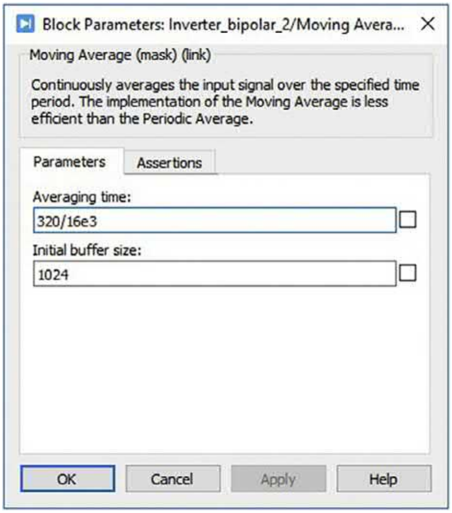
In case of a single input, all elements of the input vector are summed or subtracted. Round shape icons permit a maximum of three inputs.

Icon shape: round ☐

List of signs or number of inputs: |+ ☐

FIG. 5.96 Sum block settings.

Add a Moving Average block to output of summer in order to calculate the average of power loss (Fig. 5.92). The settings of Moving Average block are shown in Fig. 5.97. In order to obtain the steady-state waveform, the Averaging time box must be filled with a large enough number, i.e., $k \times \frac{1}{f}$, where f is switching frequency and $k > 100$ is a large number.


Block Parameters: Inverter_bipolar_2/Moving Avera...
Moving Average (mask) (link)
Parameters
Assertions
OK
Cancel
Apply
Help

Continuously averages the input signal over the specified time period. The implementation of the Moving Average is less efficient than the Periodic Average.

Averaging time: 320/16e3 ☐

Initial buffer size: 1024 ☐

FIG. 5.97 Moving average block settings.

5.14 Junction temperatures measurement

The junction temperatures can be seen with the aid of following blocks (Fig. 5.98).

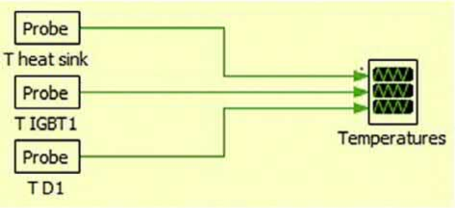
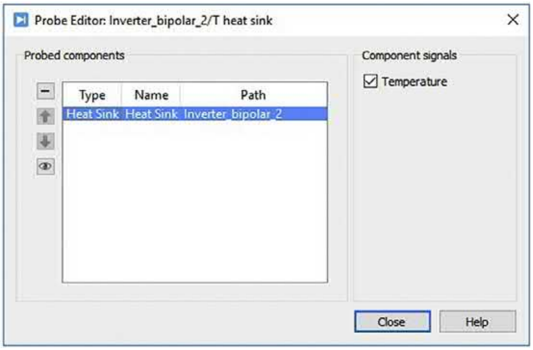


FIG. 5.98 The blocks, which monitor the junction temperatures.

The Probe block settings are shown in Figs. 5.99–5.101.


Probe Editor: Inverter_bipolar_2/T heat sink

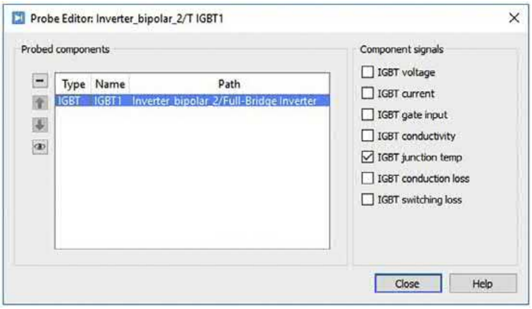
Probed components

Type	Name	Path
Heat Sink	Heat Sink	Inverter_bipolar_2

Component signals

☑ Temperature

FIG. 5.99 Drag and drop the heat sink into the T heat sink Probe block. Since the heat sink is connected to a Constant Temperature (grounded) block, its temperature keeps constant.


Probe Editor: Inverter_bipolar_2/T IGBT1

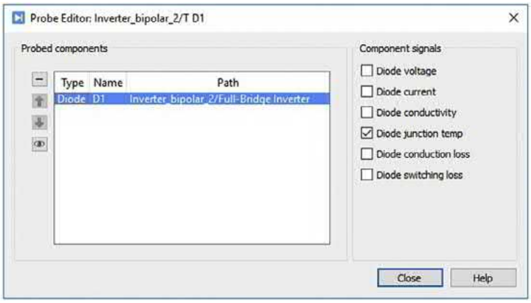
Probed components

Type	Name	Path
IGBT	IGBT1	Inverter_bipolar_2/Full-Bridge Inverter

Component signals

☐ IGBT voltage
☐ IGBT current
☐ IGBT gate input
☐ IGBT conductivity
☑ IGBT junction temp
☐ IGBT conduction loss
☐ IGBT switching loss

FIG. 5.100 Drag and drop the IGBT1 into the T IGBT1 Probe block and select the IGBT junction temp to measure the junction temperature.


Probe Editor: Inverter_bipolar_2/T D1

Probed components

Type	Name	Path
Diode	D1	Inverter_bipolar_2/Full-Bridge Inverter

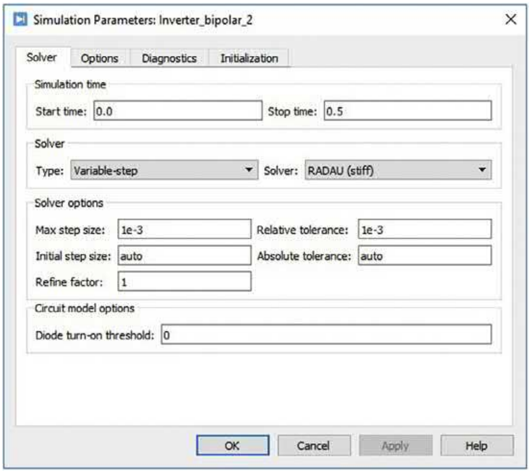
Component signals

☐ Diode voltage
☐ Diode current
☐ Diode conductivity
☑ Diode junction temp
☐ Diode conduction loss
☐ Diode switching loss

FIG. 5.101 Drag and drop the D1 into the T D1 Probe block and select the Diode junction temp to measure the junction temperature.

5.15 Running the simulation

We are ready to run the simulation. Run the simulation with the settings shown in Fig. 5.102. The simulation results are shown in Figs. 5.103–5.109.


Simulation Parameters: Inverter_bipolar_2

Simulation time

Start time: 0.0
Stop time: 0.5

Solver

Type: Variable-step
Solver: RADAU (stiff)

Solver options

Max step size: 1e-3
Relative tolerance: 1e-3
Initial step size: auto
Absolute tolerance: auto
Refine factor: 1

Circuit model options

Diode turn-on threshold: 0

FIG. 5.102 Simulation Parameters window.

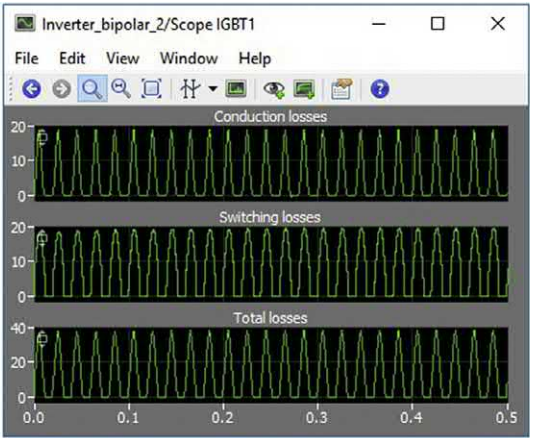


FIG. 5.103 Conduction losses, switching losses, and total losses for IGBT1.

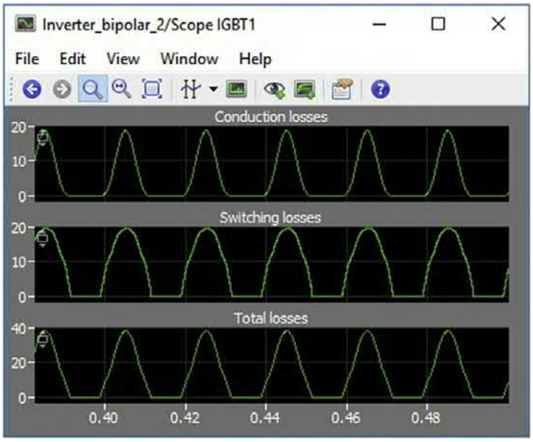

Inverter_bipolar_2/Scope IGBT1
File Edit View Window Help
Conduction losses
20
10
0
Switching losses
20
10
0
Total losses
40
20
0
0.40
0.42
0.44
0.46
0.48

FIG. 5.104 You can use the magnifier icon in order to get a closer look of waveforms shown in Fig. 5.103.

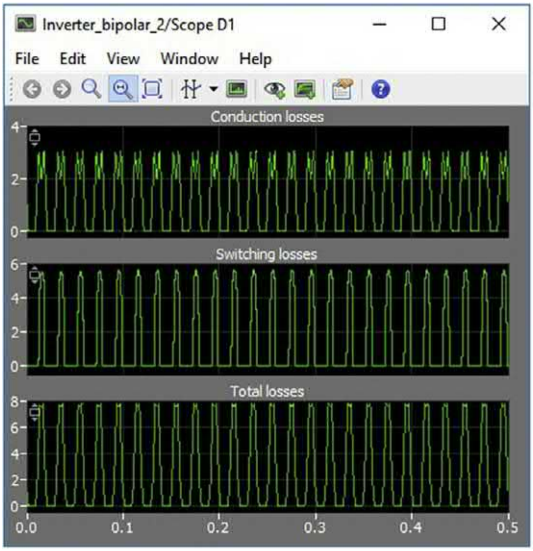

Inverter_bipolar_2/Scope D1
File Edit View Window Help
Conduction losses
4
2
0
Switching losses
6
4
2
0
Total losses
8
6
4
2
0
0.0
0.1
0.2
0.3
0.4
0.5

FIG. 5.105 Conduction losses, switching losses, and total losses for diode D1.

According to Fig. 5.109, when the heat sink temperature is 80°C, the peak junction temperature of IGBTs is less than 125°C. So, the IGBT will not fail under this condition.

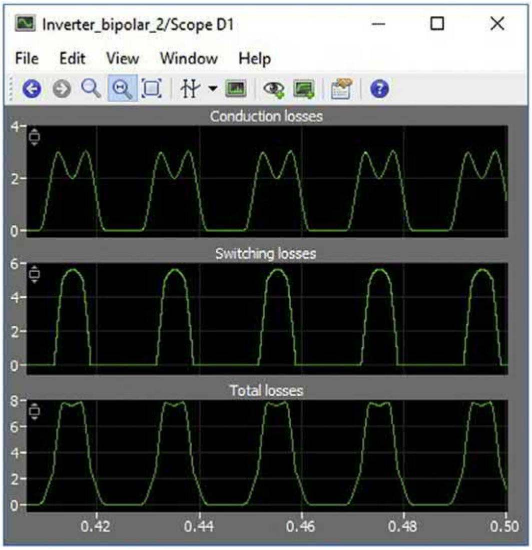

Inverter_bipolar_2/Scope D1
File Edit View Window Help
Conduction losses
4
2
0
Switching losses
6
4
2
0
Total losses
8
6
4
2
0
0.42
0.44
0.46
0.48
0.50

FIG. 5.106 You can use the magnifier icon in order to get a closer look of waveforms shown in Fig. 5.105.

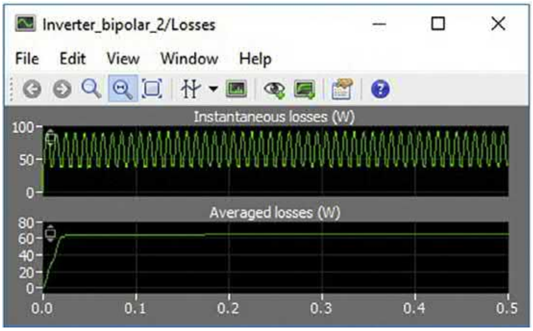

Inverter_bipolar_2/Losses
File Edit View Window Help
Instantaneous losses (W)
100
50
0
Averaged losses (W)
80
60
40
20
0
0.0
0.1
0.2
0.3
0.4
0.5

FIG. 5.107 The instantaneous and average waveform of overall total losses of semiconductor switches (IGBT1, IGBT2, IGBT3, IGBT4, D1, D2, D3, and D4).

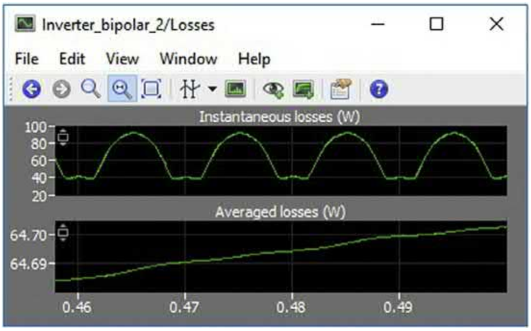

Inverter_bipolar_2/Losses
File Edit View Window Help
Instantaneous losses (W)
100
80
60
40
20
Averaged losses (W)
64.70
64.69
0.46
0.47
0.48
0.49

FIG. 5.108 A closer look to the overall total losses of semiconductor switches.

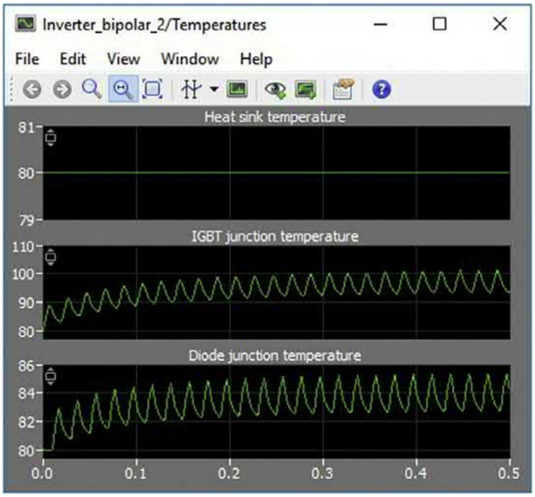

Inverter_bipolar_2/Temperatures
File Edit View Window Help
Heat sink temperature
81
80
79
IGBT junction temperature
110
100
90
80
Diode junction temperature
86
84
82
80
0.0
0.1
0.2
0.3
0.4
0.5

FIG. 5.109 Junction temperature of IGBTs and diodes.

As an exercise, you can increase the value of the Constant Temperature (Grounded) block and obtain the maximum temperature, which limit the peak junction temperature to 125°C.

5.16 Designing the heat sink

Using the measured average power (Fig. 5.107), we can calculate the thermal resistance that is required to keep the heat sink temperature at or below 80°C by using the thermal power flow equation:

$$T_{\text{heat sink}} = T_{\text{ambient}} + P_{\text{loss}}.R_{\text{th}}$$

Assume that the ambient temperature is 40°C. According to Fig. 5.107, the average power loss is about 63 W. So, the thermal resistance needed to limit the maximum heat sink temperature to 80°C is:

$$80 = 40 + 63 \times R_{\text{th}} \Longrightarrow R_{\text{th}} = 0.635 \frac{°C}{W}$$

Complete the thermal circuit by placing the calculated thermal resistance between the heat sink and ambient temperature source (Figs. 5.110, 5.111, and 5.112). Set the ambient temperature to 40°C (Fig. 5.113) and place a Heat Flow Meter block (Fig. 5.114) in series with the resistor to monitor the total thermal losses.

Set the heat sink capacitance to 10 J/K and set its initial temperature to 40°C (Fig. 5.115).

Note: The thermal capacitance is generally not calculated, but is dependent on the mass of the heat sink and the type of material. Thermal capacitance is not

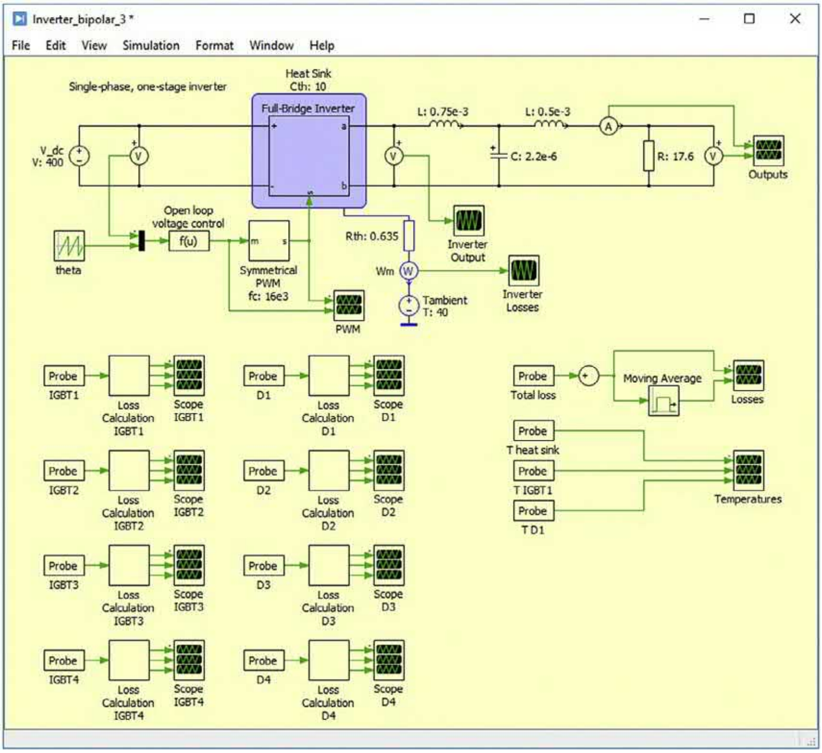

Inverter_bipolar_3 *
File
Edit
View
Simulation
Format
Window
Help
Single-phase, one-stage inverter
Heat Sink
Cth: 10
Full-Bridge Inverter
+
a
-
b
L: 0.75e-3
L: 0.5e-3
C: 2.2e-6
R: 17.6
A
V
V_dc
V: 400
Outputs
Open loop
voltage control
f(u)
m
s
theta
Symmetrical
PWM
fc: 16e3
PWM
Rth: 0.635
Wm
W
Tambient
T: 40
Inverter
Output
Inverter
Losses
Probe
IGBT1
Loss
Calculation
IGBT1
Scope
IGBT1
Probe
D1
Loss
Calculation
D1
Scope
D1
Probe
IGBT2
Loss
Calculation
IGBT2
Scope
IGBT2
Probe
D2
Loss
Calculation
D2
Scope
D2
Probe
IGBT3
Loss
Calculation
IGBT3
Scope
IGBT3
Probe
D3
Loss
Calculation
D3
Scope
D3
Probe
IGBT4
Loss
Calculation
IGBT4
Scope
IGBT4
Probe
D4
Loss
Calculation
D4
Scope
D4
Probe
Total loss
Moving Average
Losses
Probe
T heat sink
Probe
T IGBT1
Probe
T D1
Temperatures

FIG. 5.110 Thermal resistance of heat sink is considered in this added to the schematic.

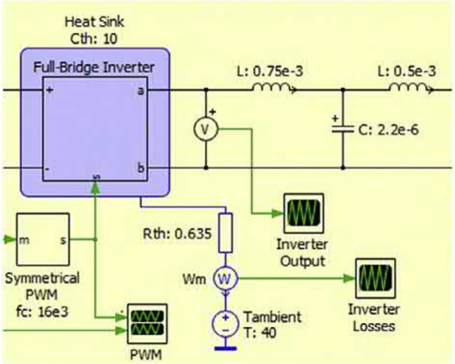
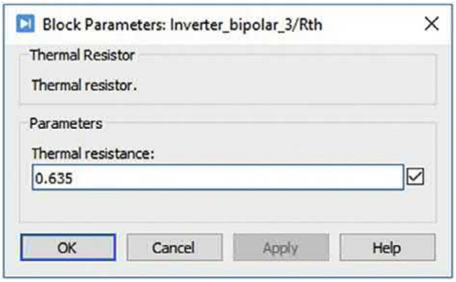

Heat Sink
Cth: 10
Full-Bridge Inverter
+
a
-
b
L: 0.75e-3
L: 0.5e-3
C: 2.2e-6
V
m
s
Symmetrical
PWM
fc: 16e3
PWM
Rth: 0.635
Wm
W
Tambient
T: 40
Inverter
Output
Inverter
Losses

FIG. 5.111 Closer look to blocks added to the schematic.



Block Parameters: Inverter_bipolar_3/Rth

Thermal Resistor

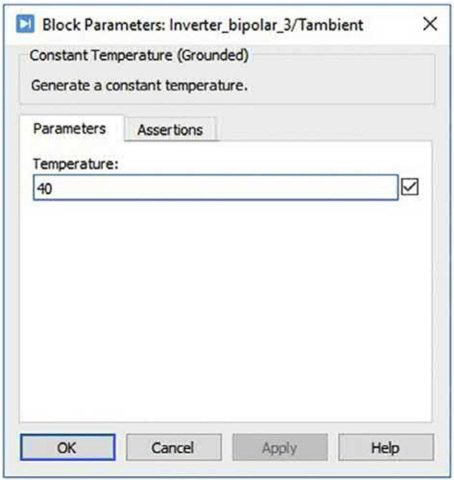
Thermal resistor.

Parameters

Thermal resistance: 0.635 ☑

OK
Cancel
Apply
Help

FIG. 5.112 Thermal resistance of heat sink is set to 0.635°C/W.


Block Parameters: Inverter_bipolar_3/Tambient

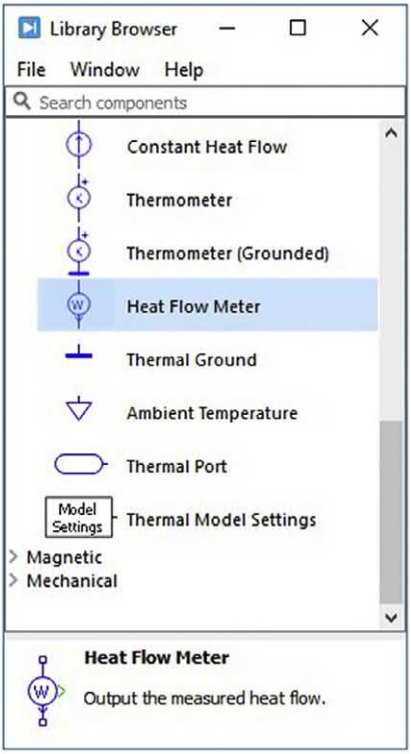
Constant Temperature (Grounded)

Generate a constant temperature.

Temperature: 40 ☑

Parameters
Assertions
OK
Cancel
Apply
Help

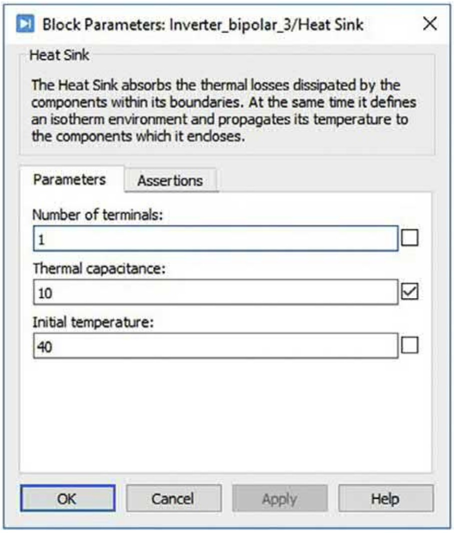
FIG. 5.113 Ambient temperature is set to 40°C.


Library Browser
File
Window
Help
Search components
Constant Heat Flow
Thermometer
Thermometer (Grounded)
Heat Flow Meter
Thermal Ground
Ambient Temperature
Thermal Port
Model Settings
Thermal Model Settings
Magnetic
Mechanical
W

Heat Flow Meter

Output the measured heat flow.

FIG. 5.114 The Heat Flow Meter block can be found in the Thermal section of Library Browser.


Block Parameters: Inverter_bipolar_3/Heat Sink

Heat Sink

The Heat Sink absorbs the thermal losses dissipated by the components within its boundaries. At the same time it defines an isotherm environment and propagates its temperature to the components which it encloses.

Number of terminals: 1 ☐

Thermal capacitance: 10 ☑

Initial temperature: 40 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 5.115 Double click the heat sink block in order to set its Thermal capacitance and Initial temperature.

as critical as thermal resistance because it is the thermal resistance that determines the steady-state operating temperature. The thermal capacitance, a measure of the heat storage ability of the heat sink, determines the rate of temperature rise during transient overload conditions.

We want to obtain the steady-state temperatures of junctions. You can use the Steady-State Analysis for this purpose. In order to do Steady-State Analysis, click the Analysis tools (Fig. 5.116).

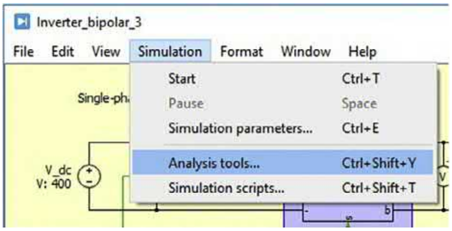

Inverter_bipolar_3
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
Single-ph
V_dc
V: 400

FIG. 5.116 Click the Analysis tools in order to do Steady-State Analysis.

The window shown in Fig. 5.117 will appear. Click the + button.

After you clicked the + button, the window shown in Fig. 5.118 will appear.

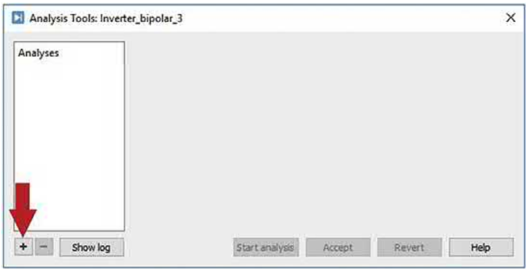
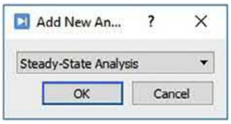

Analysis Tools: Inverter_bipolar_3
Analyses
+
−
Show log
Start analysis
Accept
Revert
Help

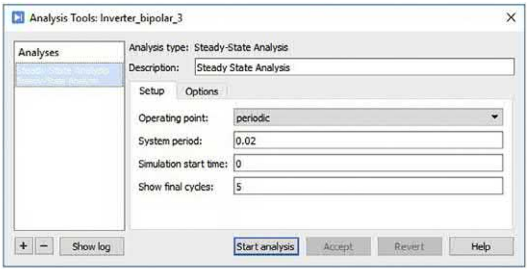
FIG. 5.117 The Analysis Tools window.


Add New An...
?
OK
Cancel

Steady-State Analysis

FIG. 5.118 Add New Analysis window.

Click the OK button. After you clicked the OK button, the Analysis Tools window will appear. Do the setting as shown in Fig. 5.119.


Analysis Tools: Inverter_bipolar_3
Analyses
Setup
Options
+
−
Show log
Start analysis
Accept
Revert
Help

Analysis type: Steady-State Analysis
Description: Steady State Analysis
Operating point: periodic
System period: 0.02
Simulation start time: 0
Show final cycles: 5

FIG. 5.119 Analysis Tools window. Since the output frequency of inverter is 50 Hz, the system period is set to 1/50 = 0.02 s.

Click the Start Analysis button in order to run the simulation. The simulation results are shown in Figs. 5.120–5.124.

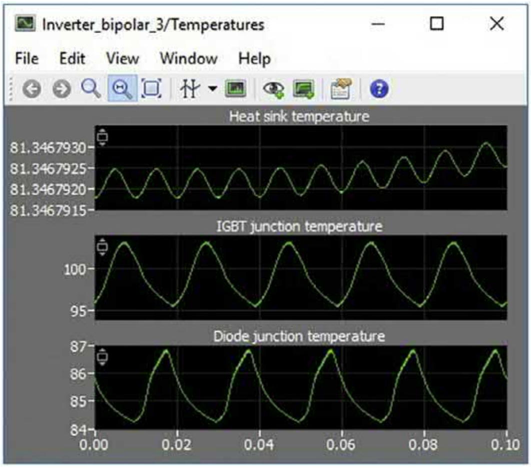

Inverter_bipolar_3/Temperatures
File
Edit
View
Window
Help
Heat sink temperature
81.3467930
81.3467925
81.3467920
81.3467915
IGBT junction temperature
100
95
Diode junction temperature
87
86
85
84
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 5.120 Temperatures of heat sink and junctions.

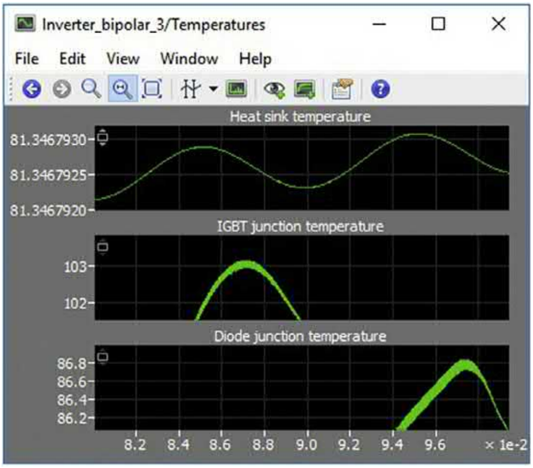

Inverter_bipolar_3/Temperatures
File
Edit
View
Window
Help
Heat sink temperature
81.3467930
81.3467925
81.3467920
IGBT junction temperature
103
102
Diode junction temperature
86.8
86.6
86.4
86.2
8.2
8.4
8.6
8.8
9.0
9.2
9.4
9.6
× 1e-2

FIG. 5.121 Closer look at Fig. 5.120. The junction temperatures are well below 125°C and the heat sink temperature is about 80°C.

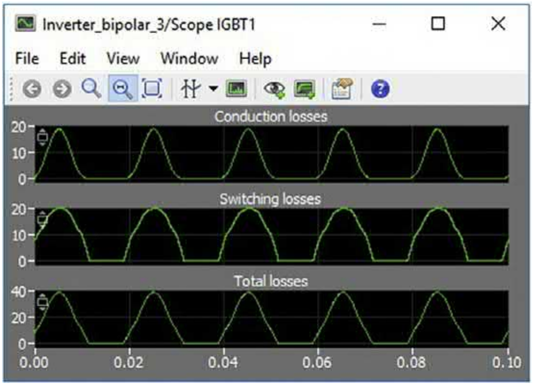

Inverter_bipolar_3/Scope IGBT1
File
Edit
View
Window
Help
Conduction losses
20
10
0
Switching losses
20
10
0
Total losses
40
20
0
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 5.122 Losses for IGBT 1.

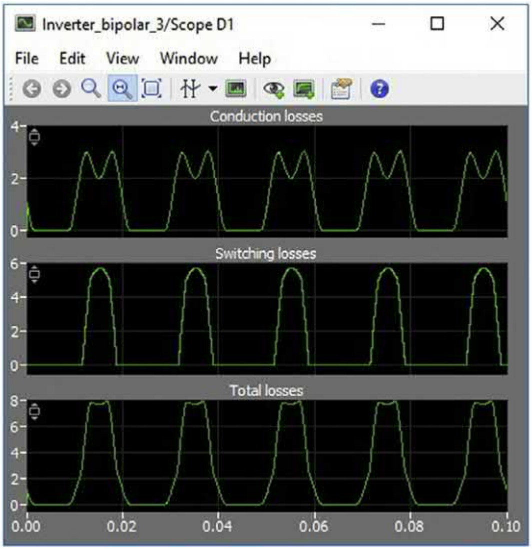

Inverter_bipolar_3/Scope D1
File Edit View Window Help
Conduction losses
4
2
0
Switching losses
6
4
2
0
Total losses
8
6
4
2
0
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 5.123 Losses for diode D1.

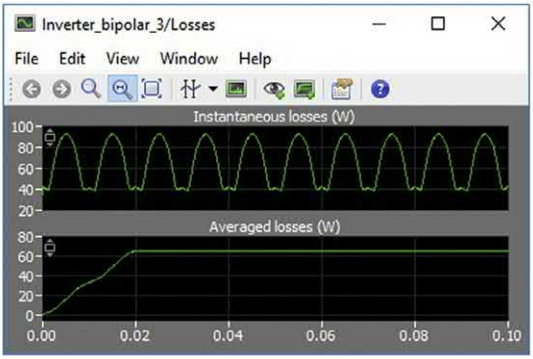

Inverter_bipolar_3/Losses
File Edit View Window Help
Instantaneous losses (W)
100
80
60
40
20
Averaged losses (W)
80
60
40
20
0
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 5.124 Graph of instantaneous and average losses.

5.17 Effect of modulation technique on losses

We used bipolar sinusoidal PWM technique in the previous sections of this chapter. In this section, we use unipolar sinusoidal PWM technique.

5.17.1 Review of unipolar PWM

With unipolar PWM, inverter legs A and B are switched separately. The switching signal for leg A is generated by comparing the modulation signal, m_a, with the triangular carrier signal. Similarly, the inverted modulation signal, m_b, is compared with the triangular carrier to generate the switching signal for leg B (Fig. 5.125).

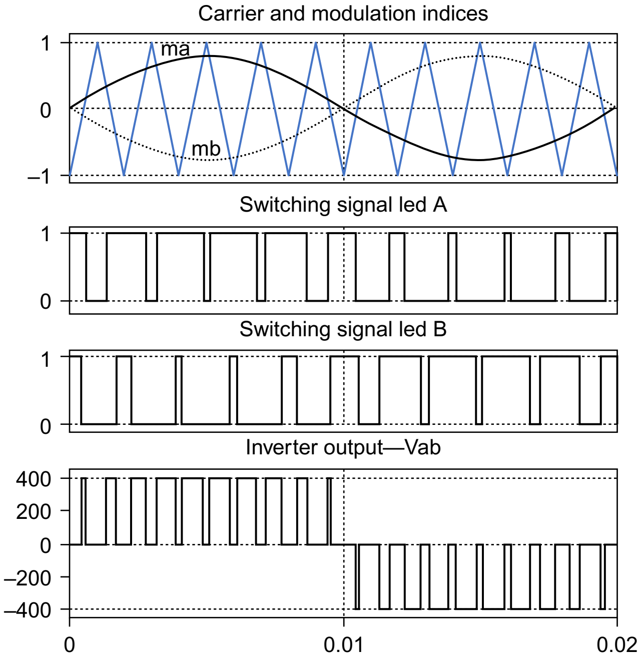


FIG. 5.125 Unipolar modulation.

Compared with bipolar PWM, unipolar PWM has an additional switching state that creates a zero-output voltage. When both bottom or top switches in legs A and B are active, each inverter leg has the same output voltage and the total inverter output voltage, V_{ab}, is therefore 0 V. With this additional switching state, the output voltage has a three-level characteristic, having a value of $\pm V_{dc}$ or 0 V.

Unipolar PWM has a number of advantages over bipolar PWM. Due to the three-level output voltage, the change in voltage across the output filter is smaller, resulting in lower current ripple. In addition, the number of IGBTs that change state during a switching event is reduced from four to two, resulting in lower switching losses. For example, when switches S2 and S4 (Fig. 5.1) are on, creating an output of 0 V, the transition to an output of V_{dc} is implemented by turning S1 on and S2 (Fig. 5.1) off. The state of S4 remains unchanged. In other words, the switch in the opposite leg retains its previous state.

Unipolar PWM does however have the drawback of generating a large common-mode voltage that can generate ground currents.

5.18 Calculation of losses for a unipolar PWM inverter

The schematic of previous bipolar inverter is changed as shown in Fig. 5.126. Only the modulation block is changed. The inside of Unipolar PWM subsystem block is shown in Fig. 5.127.

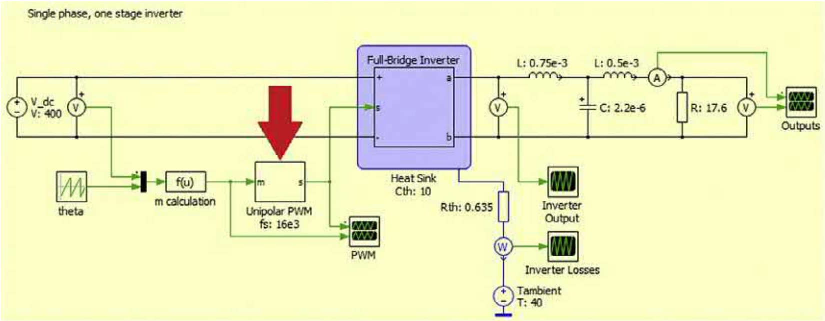

Single phase, one stage inverter
V_dc
V: 400
theta
f(u)
m calculation
m
s
Unipolar PWM
fs: 16e3
PWM
Full-Bridge Inverter
+
-
a
b
s
Heat Sink
Cth: 10
Rth: 0.635
L: 0.75e-3
L: 0.5e-3
C: 2.2e-6
R: 17.6
A
V
W
Outputs
Inverter
Output
Inverter Losses
Tambient
T: 40

FIG. 5.126 The bipolar PWM block is removed and a new subsystem is added to the schematic.

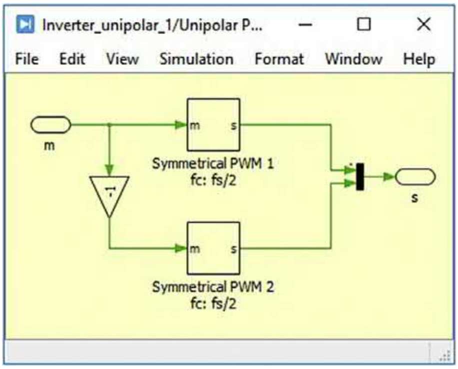

Inverter_unipolar_1/Unipolar P...
File
Edit
View
Simulation
Format
Window
Help
m
m
s
Symmetrical PWM 1
fc: fs/2
-1
m
s
Symmetrical PWM 2
fc: fs/2
s

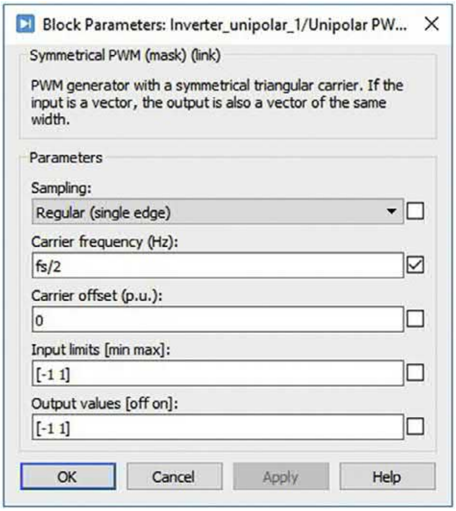
FIG. 5.127 Inside of Unipolar PWM subsystem.

The settings of Symmetrical PWM 1 and Symmetrical PWM 2 blocks (Fig. 5.127) are shown in Figs. 5.128 and 5.129, respectively.

Run the Steady-State Analysis with the settings shown in Fig. 5.130.

The analysis results are shown in Figs. 5.131 and 5.132. As shown in Fig. 5.131, the losses decreased to about 40 W. The junction temperatures are decreased accordingly.

Here is the summary of this chapter: This chapter showed how to create a thermal description for IGBTs and diodes using data sheet values. The thermal design approach of choosing a worst-case heat sink temperature as the system reference point and measuring the semiconductor losses at this operating point allows the thermal resistance required to maintain this operating point to be calculated in a single step.



Block Parameters: Inverter_unipolar_1/Unipolar PW...

Symmetrical PWM (mask) (link)

PWM generator with a symmetrical triangular carrier. If the input is a vector, the output is also a vector of the same width.

Parameters

Sampling: Regular (single edge) ☐

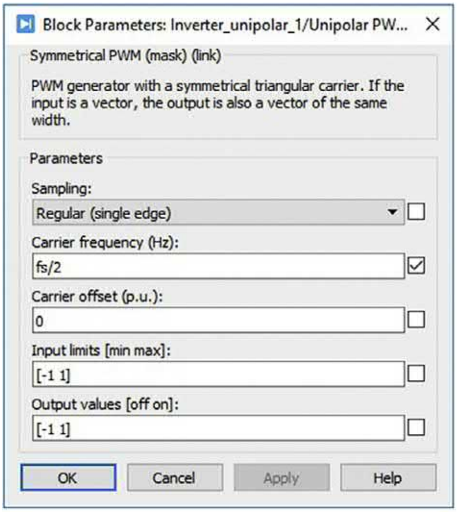
Carrier frequency (Hz): fs/2 ☑

Carrier offset (p.u.): 0 ☐

Input limits [min max]: [-1 1] ☐

Output values [off on]: [-1 1] ☐

FIG. 5.128 The Symmetrical PWM 1 (Fig. 5.127) settings, fs = 16 kHz.



Block Parameters: Inverter_unipolar_1/Unipolar PW...

Symmetrical PWM (mask) (link)

PWM generator with a symmetrical triangular carrier. If the input is a vector, the output is also a vector of the same width.

Parameters

Sampling: Regular (single edge) ☐

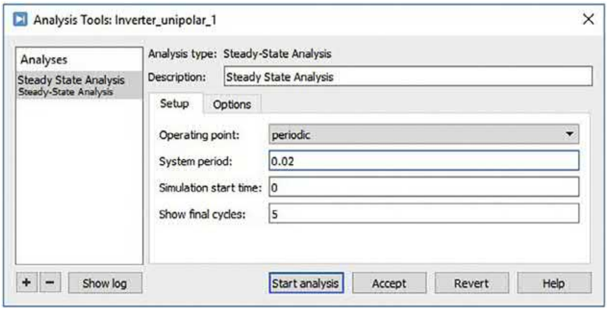
Carrier frequency (Hz): fs/2 ☑

Carrier offset (p.u.): 0 ☐

Input limits [min max]: [-1 1] ☐

Output values [off on]: [-1 1] ☐

FIG. 5.129 The Symmetrical PWM 2 (Fig. 5.127) settings, fs = 16 kHz.



Analysis Tools: Inverter_unipolar_1

Analyses

Steady State Analysis
Steady-State Analysis

Analysis type: Steady-State Analysis

Description: Steady State Analysis

Operating point: periodic

System period: 0.02

Simulation start time: 0

Show final cycles: 5

FIG. 5.130 Analysis Tools window.

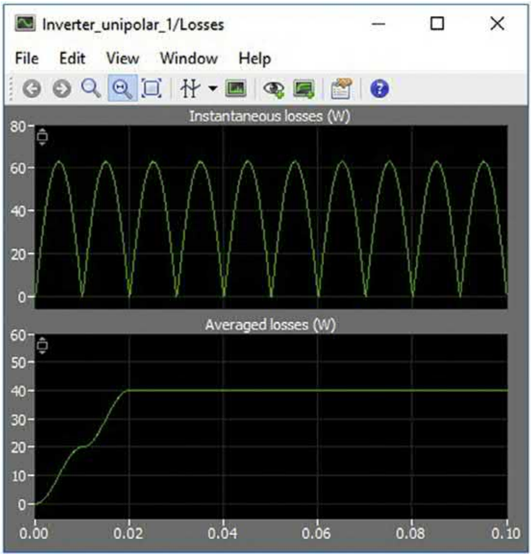

Inverter_unipolar_1/Losses
File Edit View Window Help
Instantaneous losses (W)
80
60
40
20
0
Averaged losses (W)
60
50
40
30
20
10
0
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 5.131 Graph of instantaneous and average losses.

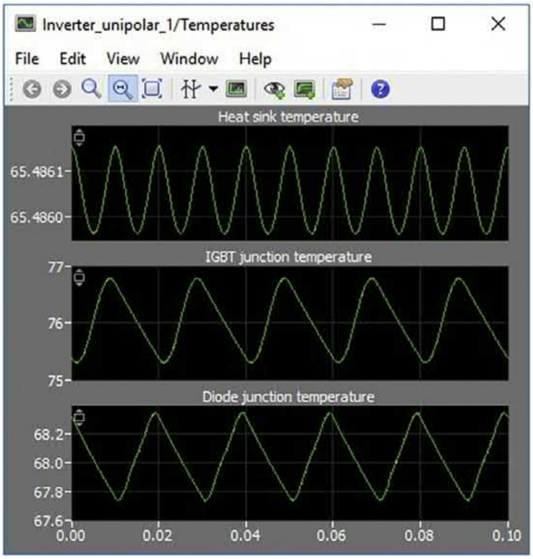

Inverter_unipolar_1/Temperatures
File Edit View Window Help
Heat sink temperature
65.4861
65.4860
IGBT junction temperature
77
76
75
Diode junction temperature
68.2
68.0
67.8
67.6
0.00
0.02
0.04
0.06
0.08
0.10

FIG. 5.132 Temperatures of heat sink and junctions.

To set the peak device junction temperature rather than the heat sink temperature to a reference point, a few design iterations are required. Thermal simulations have other uses in addition to evaluating the thermal performance of a converter or assisting with the design of a thermal circuit. They are also useful for making relative comparisons between converter losses when the modulation strategy or type of semiconductor switches is changed.

Further reading

[1] M. Marz and P. Nance, Thermal modeling of power electronic systems, www.iisb.fraunhofer.de.

[2] S. Munk-Nielsen, L.N. Tutelea, U. Jaeger, Simulation with ideal switch models combined with measured loss data provides a good estimate of power loss, in: IEEE Industry Applications Conference, vol. 5, October 8–12, 2000, pp. 2915–2922.

[3] P. Jack, Holman, Heat Transfer, McGraw-Hill Education, 2009.

[4] Y. Cengel, A. Ghajar, Heat and Mass Transfer: Fundamentals and Applications, McGraw-Hill Education, 2014.

Chapter 6

Extraction of power electronics converters uncertainties with PLECS®

Chapter outline

6.1 Introduction 365
6.2 Uncertainty models 366
  6.2.1 Parametric uncertainty 366
  6.2.2 Unstructured uncertainty 367
  6.2.3 Structured uncertainty 367
6.3 Robust control 368
  6.3.1 Kharitonov's theorem 368
  6.3.2 H_∞ Control 368
  6.3.3 μ Synthesis 369
6.4 Case study: A zeta converter 369
  6.4.1 Analyzing the system without uncertainty 370
  6.4.2 Audio susceptibility 380
  6.4.3 Output impedance 381
  6.4.4 Using the PLECS® to extract the uncertain model of the DC-DC converters 383
References 397

6.1 Introduction

Modeling is the process of formulating a mathematical description of the system. A model, no matter how detailed, is never a completely accurate representation of a real physical system. A mathematical model is always just an approximation of the true, physical reality of the system dynamics.

Uncertainty refers to the differences or errors between model and real systems and whatever methodology is used to present these errors will be called an uncertainty model. Successful robust control-system design would depend on, to a certain extent, an appropriate description of the perturbation considered.

This chapter shows how PLECS® can be used to extract the uncertain model of power electronics converters. The chapter starts with a brief review of concept of uncertainty. After reviewing the concepts, an illustrative example (a Zeta converter) is studied. In this chapter, we will use the MATLAB® programming and commands of Robust Control Toolbox®. It is assumed that the reader is familiar with MATLAB® programming.

Simulation of Power Electronics Converters Using PLECS®. https://doi.org/10.1016/B978-0-12-817364-0.00006-0
© 2020 Elsevier Inc. All rights reserved.

6.2 Uncertainty models

Different uncertainty models are studied in this section. Studying Chapters 2 and 9 of [1] is recommended in order to obtain more information about modeling uncertainties of dynamical systems.

6.2.1 Parametric uncertainty

Inaccurate description of component characteristics, torn-and-worn effects on plant components, or shifting of operating points cause dynamic perturbations in many industrial control systems. Such perturbations may be represented by variations of certain system parameters over some possible value ranges. They affect the low-frequency range performance and are called "parametric uncertainties." Studying an example is quite helpful. Assume the simple RLC circuit shown in Fig. 6.1.

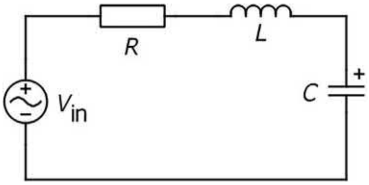


FIG. 6.1 Typical RLC circuit.

The transfer function between the capacitor voltage and the input voltage can be written as:

$$\frac{v_c(s)}{v_{in}(s)} = \frac{\frac{1}{LC}}{s^2 + \frac{R}{L}s + \frac{1}{LC}} = \frac{a}{s^2 + bs + a},$$

where $a = \frac{1}{LC}$ and $b = \frac{R}{L}$. $R, L,$ and C can be written as $R = R_0 + \delta_R, L = L_0 + \delta_L$ and $C = C_0 + \delta_C$. R_0, L_0, and C_0 show the nominal values of resistor, inductor, and capacitor, respectively. δ_R, δ_L, and δ_C show component values variations, i.e., the effect of aging, measurement error, replacement of the components, etc.

a and b can be written in the same way as, $a_0 + \delta_a$ and $b_0 + \delta_b$, respectively. $a_0 = \frac{1}{L_0C_0}$ and $b_0 = \frac{R_0}{L_0}$ show the nominal values of a and b, respectively. According to the values of R_0, L_0, C_0, δ_R, δ_L, and δ_C, a and b can be written as:

$$a_{min} < a < a_{max}$$

$$b_{min} < b < b_{max}$$

So, equation 1, no longer describe a single transfer function. It is a family of transfer functions with uncertain coefficients. It has parametric uncertainty.

6.2.2 Unstructured uncertainty

Many dynamic perturbations that may occur in different parts of a system can, however, be lumped into one single perturbation block Δ, for instance, some unmodeled, high-frequency dynamics. This uncertainty representation is referred to as "unstructured" uncertainty. In the case of linear, time-invariant systems, the block Δ may be represented by an unknown transfer function matrix. The unstructured dynamics uncertainty in a control system can be described in different ways. The most famous ones are additive and input/output multiplicative perturbation configurations. If $G_p(s)$, $G_o(s)$, and I show the perturbed system dynamics, a nominal model description of the physical system and identity matrix, respectively, then:

- Additive perturbation: $G_p(s) = G_o(s) + \Delta(s)$
- Input multiplicative perturbation: $G_p(s) = G_o(s) \times [I + \Delta(s)]$
- Output multiplicative perturbation: $G_p(s) = [I + \Delta(s)] \times G_o(s)$

In Single-Input Single-Output (SISO) systems, there is no difference between Input multiplicative perturbation and output multiplicative perturbation. In Multi-Input Multi-Output (MIMO) systems, the two descriptions are not necessarily the same.

It is a good idea to normalize the unknown transfer function Δ, i.e., $\Delta = W\Delta_n$, such that $\|\Delta_n\|_\infty \leq 1$. $\|.\|_\infty$ shows the infinity norm of the transfer function. With such a normalized Δ_n, the previous representations can be written as:

- Additive perturbation: $G_p(s) = G_o(s) + W_a(s)\Delta_n(s)$
- Input multiplicative perturbation: $G_p(s) = G_o(s) \times [I + W_m(s)\Delta_n(s)]$
- Output multiplicative perturbation: $G_p(s) = [I + W_m(s)\Delta_n(s)] \times G_o(s)$,

where $W_a(s)$ and $W_m(s)$ show the additive uncertainty weight and multiplicative uncertainty weight, respectively.

Assume that $G_{p1}(s)$, $G_{p2}(s)$, $G_{p3}(s)$, …, $G_{pn}(s)$ are n samples of perturbed system dynamics (assume that $G_{p1}(s)$, $G_{p2}(s)$, $G_{p3}(s)$, …, $G_{pn}(s)$ are SISO systems). Then, in order to calculate the additive uncertainty weight $W_a(s)$, it is enough to find such a weight in which its magnitude is greater than $G_{p1}(s) - G_o(s)$, $G_{p2}(s) - G_o(s)$, $G_{p3}(s) - G_o(s)$, …, $G_{pn}(s) - G_o(s)$ for desired frequency range.

The multiplicative uncertainty weight $W_m(s)$ can be calculated in the same way. In this case, magnitude of $W_m(s)$ must be larger than $\frac{G_{p1}(s)-G_o(s)}{G_o(s)}, \frac{G_{p2}(s)-G_o(s)}{G_o(s)}, \frac{G_{p3}(s)-G_o(s)}{G_o(s)}, \ldots, \frac{G_{pn}(s)-G_o(s)}{G_o(s)}$ in the desired frequency range.

6.2.3 Structured uncertainty

In some problems, the uncertain parts can be taken out from the dynamics and the whole system can be rearranged in a standard configuration of (upper)

Linear Fractional Transformation $F(M, \Delta)$. The uncertian block Δ would then have the following general form:

$$\Delta = \mathrm{diag}\{\delta_1 I_{r_1}, \delta_2 I_{r_2}, \delta_3 I_{r_3}, \ldots, \delta_s I_{r_s}, \Delta_1, \ldots, \Delta_f\}, \delta_i \in \mathbb{C}, \Delta_j \in \mathbb{C}^{m_j \times m_j},$$

where $\sum_{i=1}^{s} r_i + \sum_{j=1}^{f} m_j = n$ with n is the dimension of the block Δ. So, Δ consist of s repeated scalar blocks and f full blocks. The full blocks need not be square. Since the Δ considered has a certain structure, such description is called structured.

6.3 Robust control

Robust control is a design methodology that explicitly deals with uncertainty. Robust control designs a controller such that:

- Some level of performance of the controlled system is guaranteed.
- Irrespective of the changes in the plant dynamics/process dynamics within a predefined class, the stability is guaranteed.

Some of the well-known robust control design techniques are studied briefly now.

6.3.1 Kharitonov′s theorem

Kharitonov's theorem is used to assess the stability of a dynamical system when the physical parameters of the system are uncertain. It can be considered as a generalization of Routh-Hurwitz stability test. Routh-Hurwitz is concerned with an ordinary polynomial, i.e., a polynomial with fixed coefficients, while Kharitonov's theorem can study the stability of polynomials with uncertain (varying) coefficients.

Kharitonov's theorem is an analysis tool more than a synthesis tool. Kharitonov's theorem can be used to tune simple controllers such as PID. Designing high-order controllers using Kharitonov's theorem is not so common.

Refs. [2] and [3] are good references for control engineering applications of Kharitonov's theorem. Plenty of tools and related theorems are gathered there.

6.3.2 H_∞ Control

H_∞ techniques formulate the required design specifications (control goles) as an optimal control problem in the frequency domain. In order to do this, some fictitious weighting functions are added to the system model. Weighting functions are selected with respect to the required design specifications. Selection of weights is not a trivial task and requires some trial and error to obtain the desired specifications. In fact, the most crucial and difficult task in robust controller design is a choice of the weighting functions. Refs. [4–6] give very general guidelines for selection of the weights. Refs. [7] and [8] used intelligent optimization methods (genetic algorithm) to find the best weighting functions.

The H_∞ design does not always ensure robust stability and robust performance of the closed loop system. This is the main disadvantage of H_∞ design techniques.

H_∞ techniques are studied in many papers and books. Some of the well known are introduced here. Ref. [9] is the pioneering work of Zames, which introduced the H_∞ control. Ref. [10] is a good tutorial paper on H_∞ control with some numeric examples. Refs. [11] and [12] are general texts on robust control and studied the H_∞ control in detail. Refs. [13] and [14] are good references to learn how to design H_∞ controllers using MATLAB®.

6.3.3 μ Synthesis

The μ synthesis uses the D-K or μ-K iteration methods to minimize the peak value of the structured singular value of the closed-loop transfer function matrix over the set of all stabilizing controllers K. The structured singular value of a closed-loop system transfer matrix M(s), with uncertainty **Δ** and singular values σ, is defined as:

$$\|M\|_\mu = \mu_\Delta{}^{-1}(M) := \overset{min}{_{\Delta\in\Delta}}\{\overline{\sigma}(\Delta) : det(I - M\Delta) = 0\}^{\Delta\in\boldsymbol{\Delta}}$$

Usually the controller designed using the μ synthesis has a high order, which makes the implementation difficult. A model order reduction procedure is usually required.

The H_∞ control design techniques consider the system uncertainty in the unstructured form, so the controller design using the H_∞ techniques is conservative. The μ synthesis considers the uncertainty structures, so its output is less conservative.

6.4 Case study: A zeta converter

A power electronics converter is composed of passive components (i.e., capacitors and inductors), switches (i.e., MOSFET, IGBT, SCR, etc.), energy sources (i.e., Battery, AC grid, etc.), loads and some control strategy. Each component of the converter has its own variations and tolerances. Component variations come from different sources. For instance, value of a capacitor changes with temperature and aging. So, a power electronics converter can be considered as an uncertain dynamical system.

In this section, we will show how you can use PLECS® to extract the uncertain dynamical model of a Zeta converter. We will extract the nominal dynamical model first (component changes are neglected). Effect of components changes on dynamical behavior will be studied next.

You can extract the uncertain model of other types of converters in the same way shown in this chapter.

6.4.1 Analyzing the system without uncertainty

The schematic shown in Fig. 6.2 is used to extract the $\frac{v_o(s)}{d(s)}$ transfer function (i.e., the control-to-output transfer function). Nominal values of components are shown in Table 6.1.

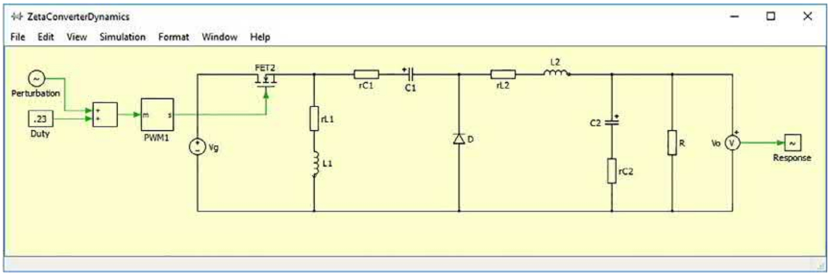

ZetaConverterDynamics
File Edit View Simulation Format Window Help
Perturbation
.23
Duty
m
s
PWM1
Vg
FET2
rL1
L1
rC1
C1
D
rL2
L2
C2
rC2
R
Vo
V
Response

FIG. 6.2 Extraction of the $\frac{v_o(s)}{d(s)}$ transfer function Bode diagram for the studied Zeta converter.

TABLE 6.1 Nominal values of Zeta converter parameters (see Fig. 6.2).

	Nominal value
Input DC source voltage, Vg	20 V
MOSFET Drain-Source resistance, rds	10 mΩ
Capacitor, C_1	100 μF
Capacitor C_1 Equivalent Series Resistance (ESR), rC_1	0.19 Ω
Capacitor, C_2	220 μF
Capacitor C_2 Equivalent Series Resistance (ESR), rC_2	0.095 Ω
Inductor, L_1	100 μH
Inductor ESR, rL_1	1 mΩ
Inductor, L_2	55 μH
Inductor ESR, rL_2	0.55 mΩ
Diode voltage drop, vD	0.7 V
Diode forward resistance, rD	10 mΩ
Load resistor, R	6 Ω
Switching Frequency, Fsw	100 Hz
Duty ratio	0.23

The "Small Signal Perturbation" and "Small Signal Response" blocks (Fig. 6.3) are added to suitable places in order to extract the small signal transfer function from perturbation of duty ratio to the perturbation of the output voltage.

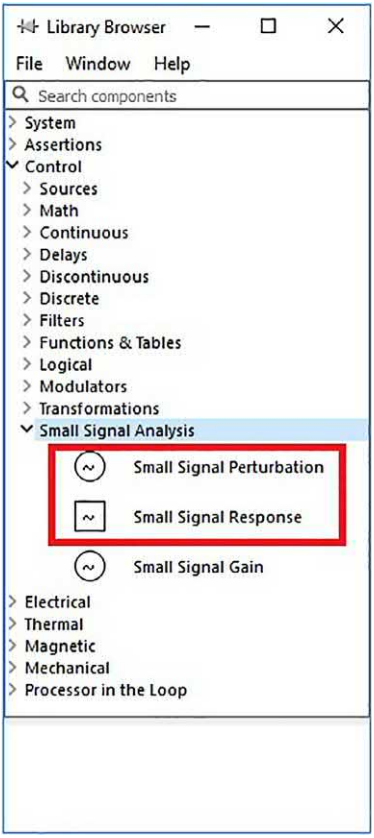
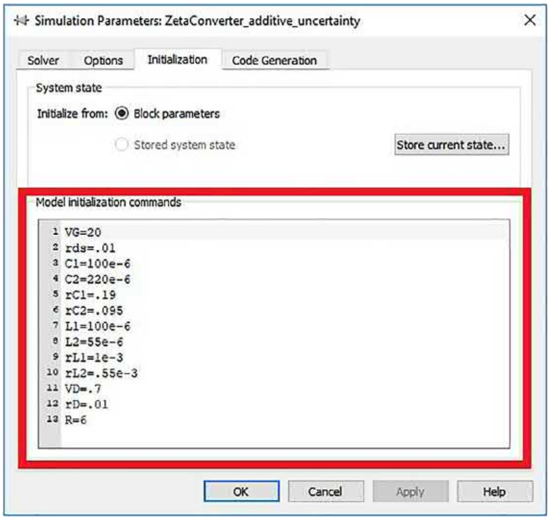

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Transformations
Small Signal Analysis
Small Signal Perturbation
Small Signal Response
Small Signal Gain
Electrical
Thermal
Magnetic
Mechanical
Processor in the Loop

FIG. 6.3 "Small Signal Perturbation" and "Small Signal Response" blocks.

You can double click each component and enter its value like the previous chapters. However, use of Initialization tab of Simulation Parameters window (Fig. 6.4) is recommended in this chapter. As you will see, this simplifies the tasks considerably.


Simulation Parameters: ZetaConverter_additive_uncertainty
Solver
Options
Initialization
Code Generation
Store current state...
OK
Cancel
Apply
Help

System state

Initialize from: ● Block parameters

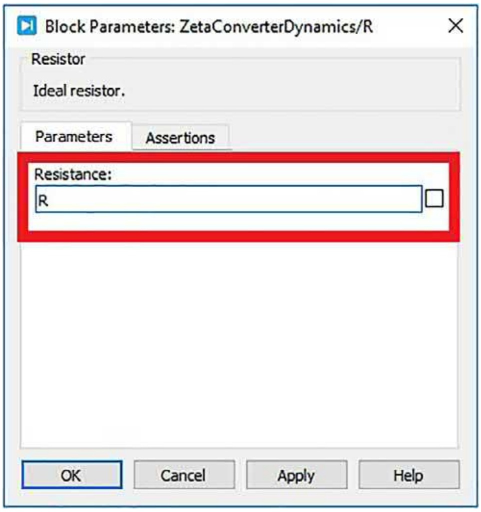
○ Stored system state

Model initialization commands

```
VG=20
rds=.01
C1=100e-6
C2=220e-6
rC1=.19
rC2=.095
L1=100e-6
L2=55e-6
rL1=1e-3
rL2=.55e-3
VD=.7
rD=.01
R=6
```

FIG. 6.4 Simulation Parameters window appear when you click the Simulation Parameters … in the Simulation menu. rds,VD, and rD show the Drain-Source resistance of closed MOSFET, voltage drop of forward-biased diode, and series resistance of forward-biased diode, respectively.

After variables are defined in the Model initialization commands section of Simulation Parameters window (Fig. 6.4), you must double click each component inside the schematic editor and enter the corresponding variable into it. For instance, variable named R is used to define the output load value. You must double click the output load and write R in its resistance box in order to tell PLECS® that R is used for output load (Fig. 6.5).


Block Parameters: ZetaConverterDynamics/R
Parameters
Assertions
OK
Cancel
Apply
Help

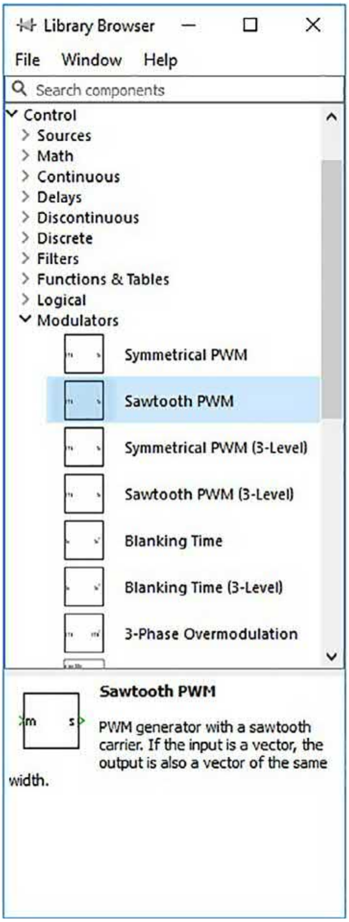
Resistor

Ideal resistor.

Resistance: R ☐

FIG. 6.5 Double click the output load and write R in the Resistance box in order to tell PLECS® that variable R is used to represent the output load. Variable R value is 6 Ω (see Fig. 6.4).

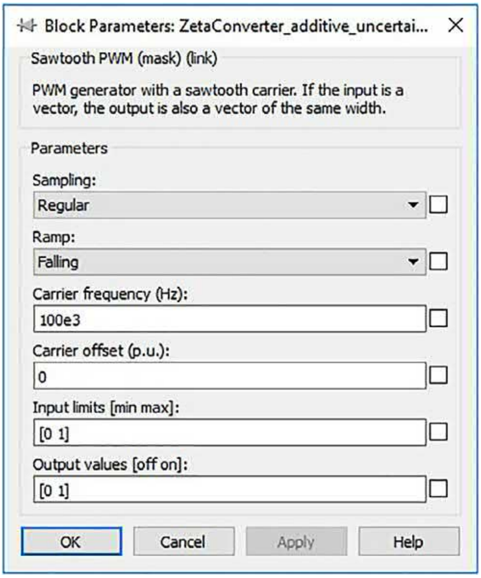
The PWM signal is produced with the aid of a "Sawtooth PWM" block (Fig. 6.6). Fig. 6.7 shows the settings of the "Sawtooth PWM" block used in Fig. 6.2.


Library Browser
File
Window
Help
Search components
Control
Sources
Math
Continuous
Delays
Discontinuous
Discrete
Filters
Functions & Tables
Logical
Modulators
Symmetrical PWM
Sawtooth PWM
Symmetrical PWM (3-Level)
Sawtooth PWM (3-Level)
Blanking Time
Blanking Time (3-Level)
3-Phase Overmodulation
m
s

Sawtooth PWM

PWM generator with a sawtooth carrier. If the input is a vector, the output is also a vector of the same width.

FIG. 6.6 "Sawtooth PWM" block.



Block Parameters: ZetaConverter_additive_uncertai...

Sawtooth PWM (mask) (link)

PWM generator with a sawtooth carrier. If the input is a vector, the output is also a vector of the same width.

Parameters

Sampling: Regular ☐

Ramp: Falling ☐

Carrier frequency (Hz): 100e3 ☐

Carrier offset (p.u.): 0 ☐

Input limits [min max]: [0 1] ☐

Output values [off on]: [0 1] ☐

OK
Cancel
Apply
Help

FIG. 6.7 Settings of the "Sawtooth PWM" block used in Fig. 6.2.

Click the "Analysis tools…" in order to obtain the converter small signal transfer function (Figs. 6.8 and 6.9).

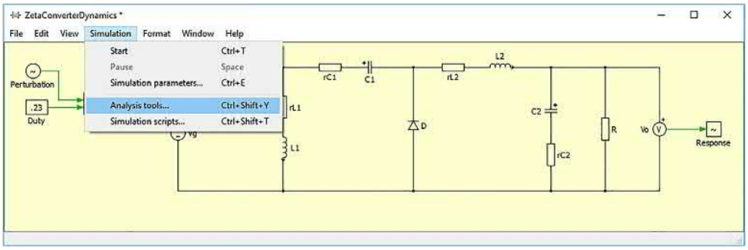
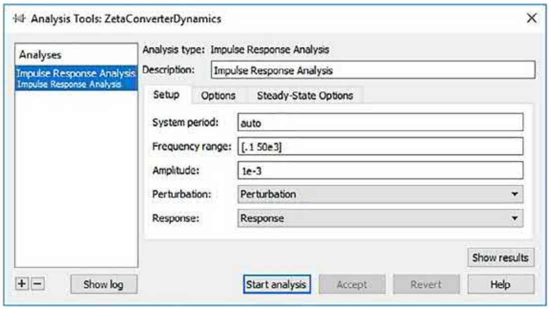

ZetaConverterDynamics *
File
Edit
View
Simulation
Format
Window
Help
Start Ctrl+T
Pause Space
Simulation parameters... Ctrl+E
Analysis tools... Ctrl+Shift+Y
Simulation scripts... Ctrl+Shift+T
Perturbation
.23
Duty
Vg
rL1
L1
rC1
C1
D
rL2
L2
C2
rC2
R
Vo
V
Response

FIG. 6.8 Click "Analysis tools…" in order to open the "Analysis Tools" window.



Analysis Tools: ZetaConverterDynamics

Analyses: Impulse Response Analysis

Analysis type: Impulse Response Analysis

Description: Impulse Response Analysis

Setup | Options | Steady-State Options

System period: auto

Frequency range: [.1 50e3]

Amplitude: 1e-3

Perturbation: Perturbation

Response: Response

Show results
+
−
Show log
Start analysis
Accept
Revert
Help

FIG. 6.9 The desired frequency range is entered into the "Frequency range" box. The "Amplitude" box is filled with a small number. The perturbation produced by the Perturbation block will be added to the duty cycle which has the steady-state value of 0.23 (see Table 6.1), so, the amplitude of Perturbation block must be quite small in comparison to 0.23.

Enter the desired frequency range in the opened window and click the "Start analysis" button.

The simulation result is shown in Fig. 6.10.

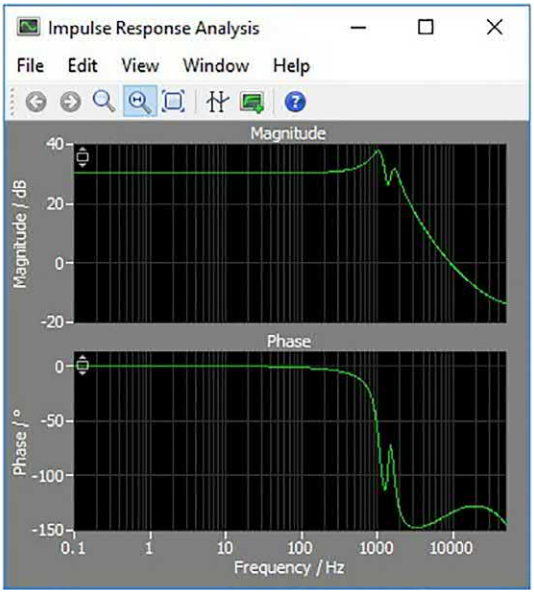


FIG. 6.10 Bode diagram of $\frac{v_o(s)}{d(s)}$ transfer function.

You can export the obtained result as a .csv (Comma Separated Values) file. The .csv file is the medium to transfer the obtained results into MATLAB®. Use the File menu in order to export the graphics as a .csv file (Fig. 6.11).

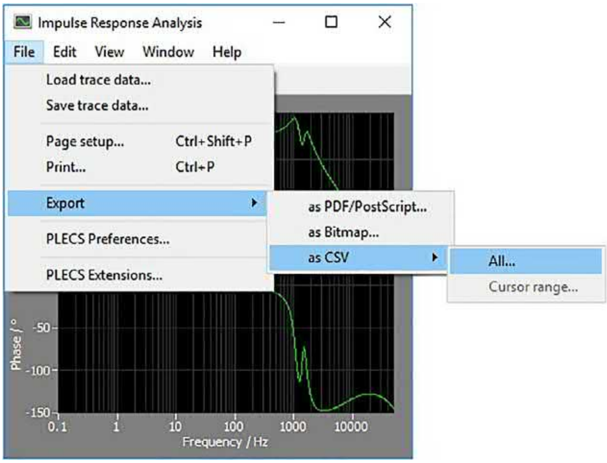
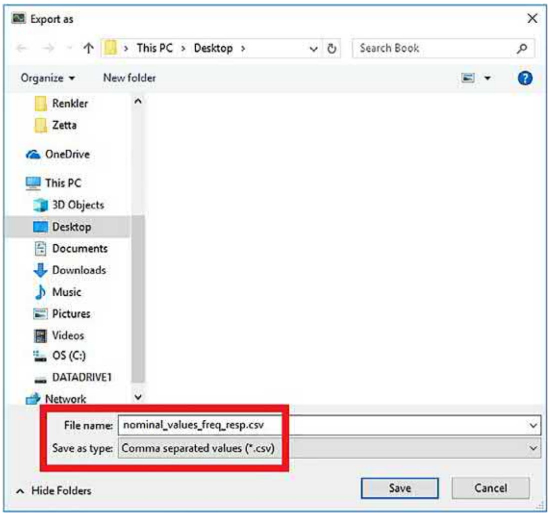

Impulse Response Analysis
File
Edit
View
Window
Help
Load trace data...
Save trace data...
Page setup... Ctrl+Shift+P
Print... Ctrl+P
Export
PLECS Preferences...
PLECS Extensions...
as PDF/PostScript...
as Bitmap...
as CSV
All...
Cursor range...
Phase / °
-50
-100
-150
0.1
1
10
100
1000
10000
Frequency / Hz

FIG. 6.11 Use "Export" to produce the .csv files.

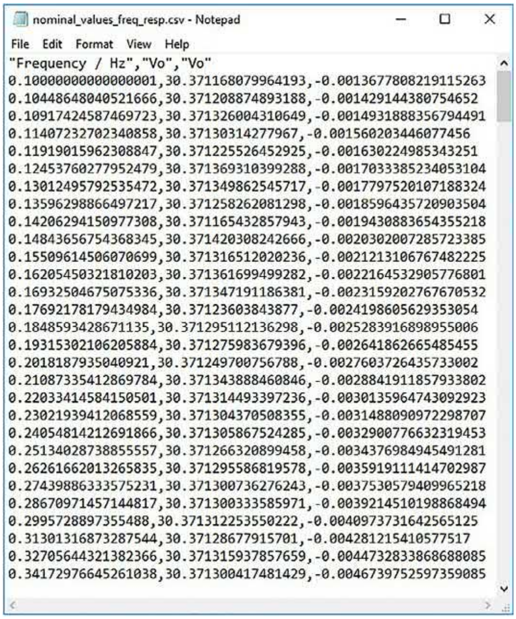
Assign the desired name and path in the "Export as" window. Here, the "nominal_values_freq_resp.csv" name is selected (Fig. 6.12).



File name: nominal_values_freq_resp.csv
Save as type: Comma separated values (*.csv)

FIG. 6.12 The "Export as" window.

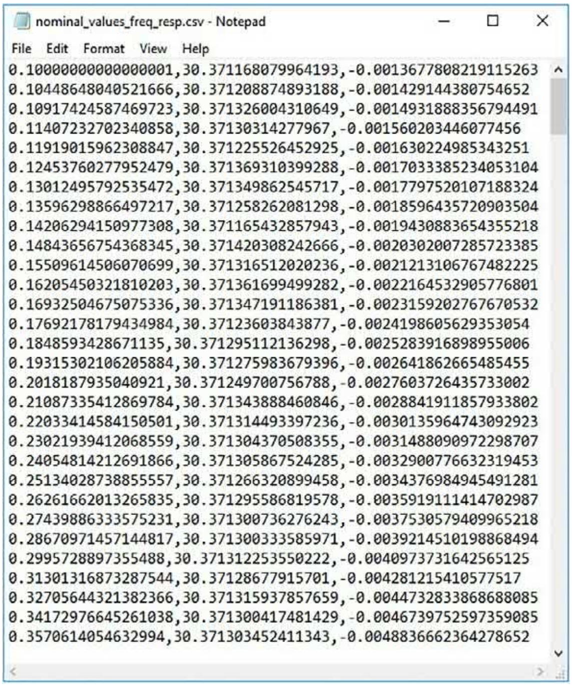
You can open the saved file (nominal_values_freq_resp.csv) in Notepad. As shown in Fig. 6.13, the saved file has three columns. First column is the frequency in Hertz, second column is the magnitude in dB, and finally the third one is the phase in Radians.



```
"Frequency / Hz","Vo","Vo"
0.10000000000000001,30.371168079964193,-0.0013677808219115263
0.10448648040521666,30.371208874893188,-0.001429144380754652
0.10917424587469723,30.371326004310649,-0.0014931888356794491
0.11407232702340858,30.37130314277967,-0.001560203446077456
0.11919015962308847,30.371225526452925,-0.001630224985343251
0.12453760277952479,30.371369310399288,-0.0017033385234053104
0.13012495792535472,30.371349862545717,-0.0017797520107188324
0.13596298866497217,30.371258262081298,-0.0018596435720903504
0.14206294150977308,30.371165432857943,-0.0019430883654355218
0.14843656754368345,30.371420308242666,-0.0020302007285723385
0.15509614506070699,30.371316512020236,-0.0021213106767482225
0.16205450321810203,30.371361699499282,-0.0022164532905776801
0.16932504675075336,30.371347191186381,-0.0023159202767670532
0.17692178179434984,30.37123603843877,-0.0024198605629353054
0.1848593428671135,30.371295112136298,-0.0025283916898955006
0.19315302106205884,30.371275983679396,-0.002641862665485455
0.2018187935040921,30.371249700756788,-0.0027603726435733002
0.21087335412869784,30.371343888460846,-0.0028841911857933802
0.22033414584150501,30.371314493397236,-0.0030135964743092923
0.23021939412068559,30.371304370508355,-0.0031488090972298707
0.24054814212691866,30.371305867524285,-0.0032900776632319453
0.25134028738855557,30.371266320899458,-0.0034376984945491281
0.26261662013265835,30.371295586819578,-0.0035919111414702987
0.27439886333575231,30.371300736276243,-0.0037530579409965218
0.28670971457144817,30.371300333585971,-0.0039214510198868494
0.2995728897355488,30.371312253550222,-0.0040973731642565125
0.31301316873287544,30.37128677915701,-0.004281215410577517
0.32705644321382366,30.371315937857659,-0.0044732833868688085
0.34172976645261038,30.371300417481429,-0.0046739752597359085
```

FIG. 6.13 The .csv is opened in Notepad.

Clear the first line ("Frequency /Hz,""Vo," "Vo") before importing the file into the MATLAB® environment (Fig. 6.14).

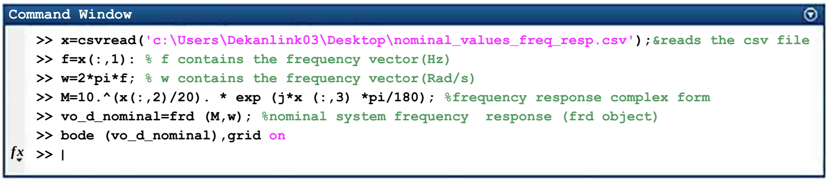


nominal_values_freq_resp.csv - Notepad
File Edit Format View Help

```
0.10000000000000001,30.371168079964193,-0.0013677808219115263
0.10448648040521666,30.371208874893188,-0.001429144380754652
0.10917424587469723,30.371326004310649,-0.0014931888356794491
0.11407232702340858,30.37130314277967,-0.001560203446077456
0.11919015962308847,30.371225526452925,-0.001630224985343251
0.12453760277952479,30.371369310399288,-0.0017033385234053104
0.13012495792535472,30.371349862545717,-0.0017797520107188324
0.13596298866497217,30.371258262081298,-0.0018596435720903504
0.14206294150977308,30.371165432857943,-0.0019430883654355218
0.14843656754368345,30.371420308242666,-0.0020302007285723385
0.15509614506070699,30.371316512020236,-0.0021213106767482225
0.16205450321810203,30.371361699499282,-0.0022164532905776801
0.16932504675075336,30.371347191186381,-0.0023159202767670532
0.17692178179434984,30.37123603843877,-0.0024198605629353054
0.1848593428671135,30.371295112136298,-0.0025283916898955006
0.19315302106205884,30.371275983679396,-0.002641862665485455
0.2018187935040921,30.371249700756788,-0.0027603726435733002
0.21087335412869784,30.371343888460846,-0.0028841911857933802
0.22033414584150501,30.371314493397236,-0.0030135964743092923
0.23021939412068559,30.371304370508355,-0.0031488090972298707
0.24054814212691866,30.371305867524285,-0.0032900776632319453
0.25134028738855557,30.371266320899458,-0.0034376984945491281
0.26261662013265835,30.371295586819578,-0.0035919111414702987
0.27439886333575231,30.371300736276243,-0.0037530579409965218
0.28670971457144817,30.371300333585971,-0.0039214510198868494
0.2995728897355488,30.371312253550222,-0.0040973731642565125
0.31301316873287544,30.37128677915701,-0.004281215410577517
0.32705644321382366,30.371315937857659,-0.0044732833868688085
0.34172976645261038,30.371300417481429,-0.0046739752597359085
0.3570614054632994,30.371303452411343,-0.0048836662364278652
```

FIG. 6.14 First line of the .csv file is removed.

The following code (Fig. 6.15) reads the .csv file and redraws the Bode diagram in the MATLAB® environment. The drawn Bode diagram is shown in Fig. 6.16. This figure is the same as Fig. 6.10. So, we import the results into the MATLAB® successfully. Using the "`tfest`" command, one can estimate a transfer function for the diagram provided by PLECS®.



```
>> x=csvread('c:\Users\Dekanlink03\Desktop\nominal_values_freq_resp.csv');&reads the csv file
>> f=x(:,1): % f contains the frequency vector(Hz)
>> w=2*pi*f; % w contains the frequency vector(Rad/s)
>> M=10.^(x(:,2)/20). * exp (j*x (:,3) *pi/180); %frequency response complex form
>> vo_d_nominal=frd (M,w); %nominal system frequency  response (frd object)
>> bode (vo_d_nominal),grid on
>> |
```

FIG. 6.15 Required code for importing the produced .csv file into MATLAB®.

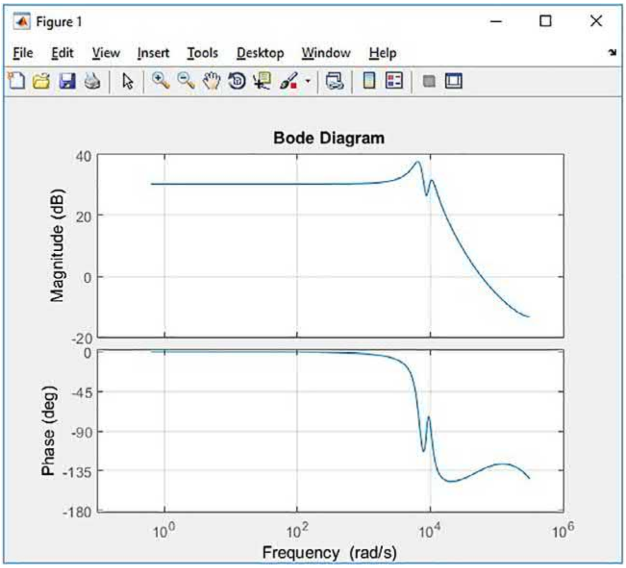


FIG. 6.16 Fig. 6.10 is regenerated in MATLAB® environment. The horizontal axis in Fig. 6.10 uses Hz unit, while this figure uses $\frac{rad}{s}$.

You can use the inductor currents as output as well. For instance, the schematic shown in Fig. 6.17 extracts the $\frac{i_{L_2}(s)}{d(s)}$ transfer function. The "Am1" is an ammeter block. Simulation results are shown in Fig. 6.18.

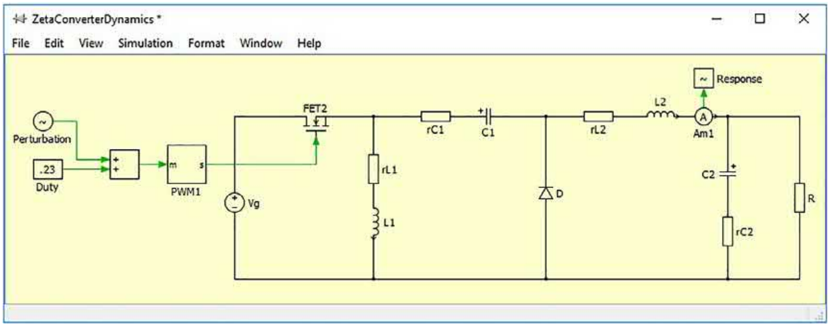

ZetaConverterDynamics *
File Edit View Simulation Format Window Help
Response
Perturbation
.23
Duty
PWM1
m
s
FET2
Vg
rL1
L1
rC1
C1
D
rL2
L2
Am1
C2
rC2
R

FIG. 6.17 Extraction of the $\frac{i_{L_2}(s)}{d(s)}$ transfer function Bode diagram.

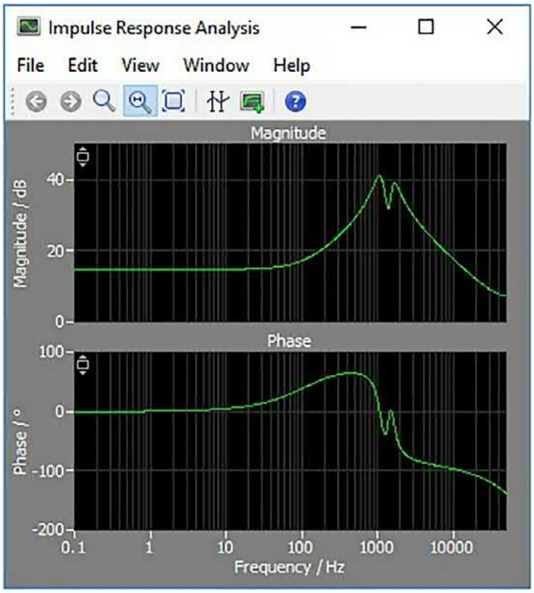
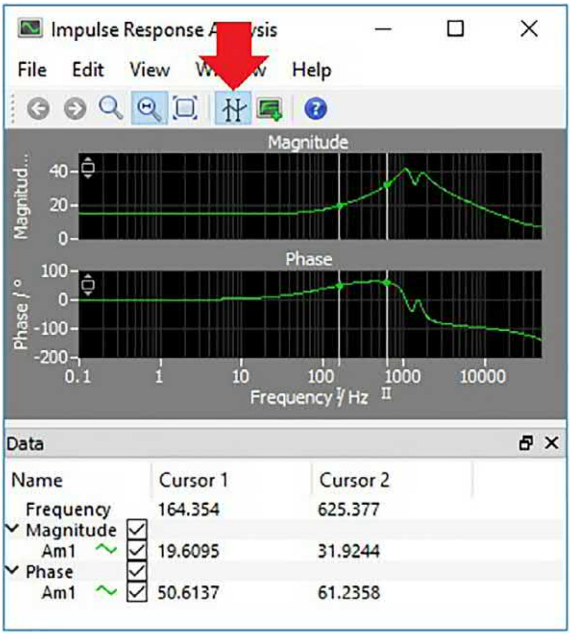

Impulse Response Analysis
File
Edit
View
Window
Help
Magnitude
Magnitude / dB
40
20
0
Phase
100
0
-100
-200
Phase / °
0.1
1
10
100
1000
10000
Frequency / Hz

FIG. 6.18 Bode diagram of $\frac{i_{L_2}(s)}{d(s)}$ transfer function.

You can add cursor to the shown Bode diagram using the icon shown in Fig. 6.19.


Impulse Response Analysis
File
Edit
View
Window
Help
Magnitude
Magnitud...
40
20
0
Phase
100
0
-100
-200
Phase / °
0.1
1
10
100
1000
10000
Frequency / Hz
I
II
Data

Name	Cursor 1	Cursor 2
Frequency	164.354	625.377
Magnitude ☑		
Am1 ☑	19.6095	31.9244
Phase ☑		
Am1 ☑	50.6137	61.2358

FIG. 6.19 Cursors can be used to read the diagram easily. Use the Cursors icon (shown with arrow) to add cursors to the diagram.

6.4.2 Audio susceptibility

The open-loop line-to-output transfer function (also termed power supply ripple rejection (PSRR) or audio susceptibility) is defined as the transfer function from perturbation of the input voltage to perturbation of the output voltage with duty ratio held constant. The audio susceptibility of the studied Zeta converter can be obtained with the aid of the schematic shown in Fig. 6.20. The Perturbation block adds small perturbations to the input voltage and the Response block measures the effects on output voltage.

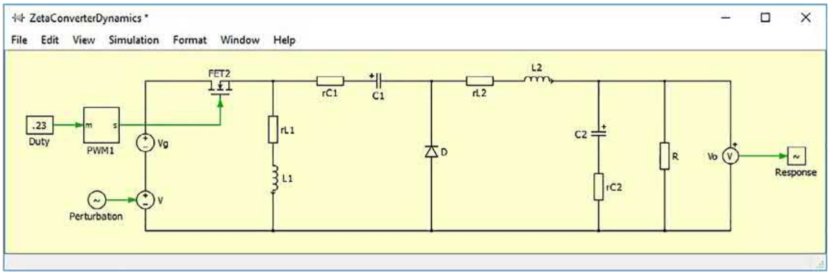
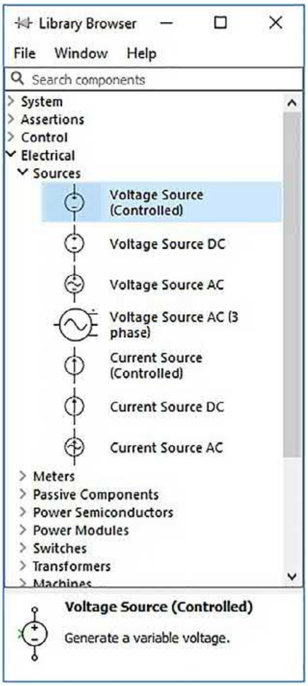

ZetaConverterDynamics *
File
Edit
View
Simulation
Format
Window
Help
FET2
rC1
C1
rL2
L2
.23
Duty
m
s
PWM1
Vg
rL1
L1
D
C2
rC2
R
Vo
V
Response
V
Perturbation

FIG. 6.20 Extraction of the $\frac{v_o(s)}{v_{in}(s)}$ transfer function.

The block named "V" in the schematic shown in Fig. 6.21 is a controlled voltage source.

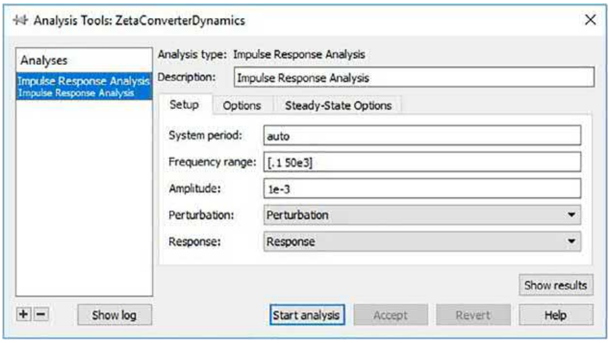

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Sources
Voltage Source (Controlled)
Voltage Source DC
Voltage Source AC
Voltage Source AC (3 phase)
Current Source (Controlled)
Current Source DC
Current Source AC
Meters
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines

Voltage Source (Controlled)

Generate a variable voltage.

FIG. 6.21 Controlled voltage source block can be found in the Sources section of Library Browser.

After running the simulation with the settings shown in Fig. 6.22, the result shown in Fig. 6.23 is obtained.


Analysis Tools: ZetaConverterDynamics
Analyses
Impulse Response Analysis
Setup
Options
Steady-State Options
Show results
Show log
Start analysis
Accept
Revert
Help

Analysis type: Impulse Response Analysis
Description: Impulse Response Analysis
System period: auto
Frequency range: [.1 50e3]
Amplitude: 1e-3
Perturbation: Perturbation
Response: Response

FIG. 6.22 Simulation settings.

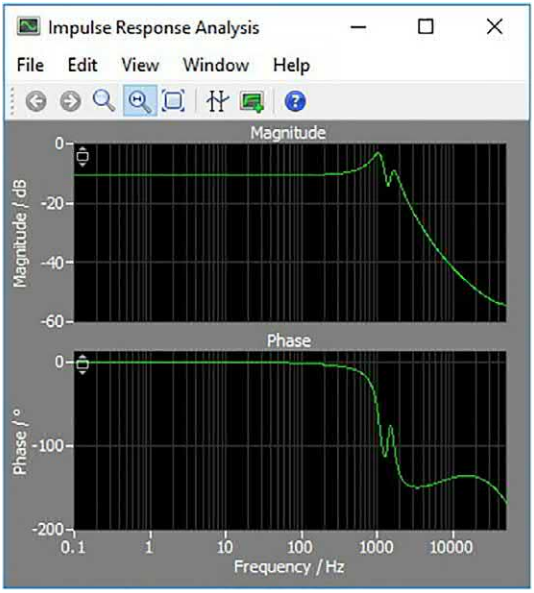


FIG. 6.23 Bode diagram of $\frac{v_o(s)}{v_{in}(s)}$ transfer function.

6.4.3 Output impedance

The converter output impedance can be extracted with the aid of schematic shown in Fig. 6.24. The Perturbation block injects a small current to the output of converter and the Response block measures the effects on output voltage. So, result of analysis has the dimension of Ohm's (since $\frac{changes\ of\ output\ voltage\ (V)}{changes\ of\ injected\ current\ (A)}$ has the dimension of Ohm's).

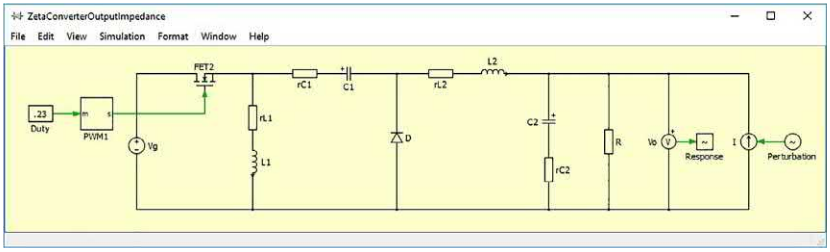
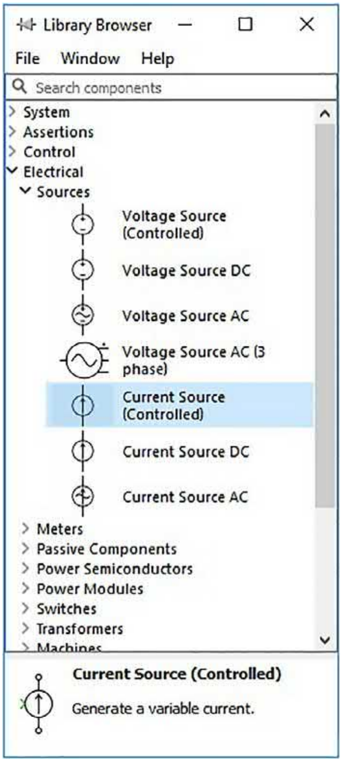

ZetaConverterOutputImpedance
File Edit View Simulation Format Window Help
FET2
rC1
C1
rL2
L2
.23
Duty
m
s
PWM1
Vg
rL1
L1
D
C2
rC2
R
Vo
V
Response
I
Perturbation

FIG. 6.24 Extraction of output impedance.

The block named "I" in the schematic shown in Fig. 6.25 is a controlled current source.

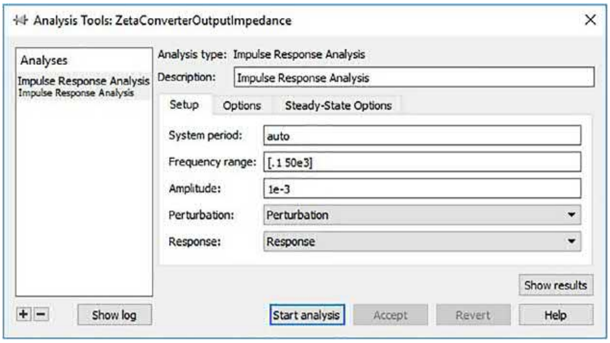

Library Browser
File Window Help
Search components
System
Assertions
Control
Electrical
Sources
Voltage Source (Controlled)
Voltage Source DC
Voltage Source AC
Voltage Source AC (3 phase)
Current Source (Controlled)
Current Source DC
Current Source AC
Meters
Passive Components
Power Semiconductors
Power Modules
Switches
Transformers
Machines

Current Source (Controlled)

Generate a variable current.

FIG. 6.25 Controlled current source block can be found in the Sources section of Library Browser.

After running the simulation with the settings shown in Fig. 6.26, the result shown in Fig. 6.27 is obtained.


Analysis Tools: ZetaConverterOutputImpedance

Analyses
Impulse Response Analysis
Impulse Response Analysis

Analysis type: Impulse Response Analysis
Description: Impulse Response Analysis

System period: auto
Frequency range: [.1 50e3]
Amplitude: 1e-3
Perturbation: Perturbation
Response: Response

FIG. 6.26 Simulation settings.

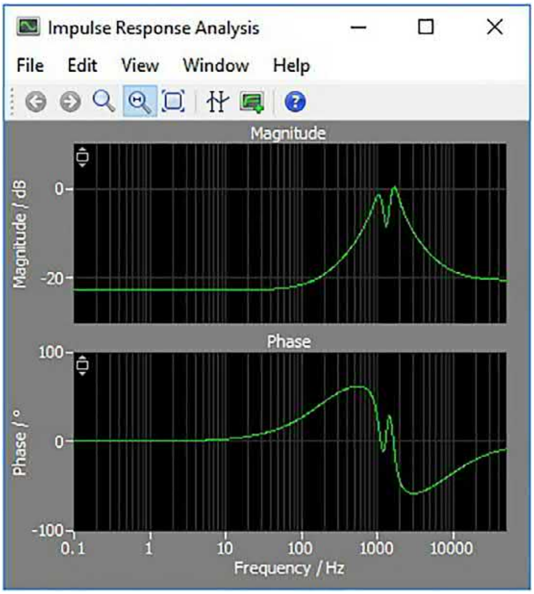


FIG. 6.27 Bode diagram of output impedance.

6.4.4 Using the PLECS® to extract the uncertain model of the DC-DC converters

The previous section studied the small signal transfer functions of the converter for nominal values of components. We neglect the component changes in the previous section of this chapter. In this section, we consider the components changes. The components changes come from different sources as mentioned before. For instance, each component has its own tolerances. Even aging and temperature changes cause the component values to change. In this section, effects of components changes on the converter dynamics are studied. This section shows how PLECS® can be used to extract the uncertain model of studied Zeta converter.

6.4.4.1 *Additive uncertainty model*

Assume that the Zeta converter components have variations shown in Table 6.2.

For instance, according to Table 6.2, ESR of C_1 changes between 0.19 $\Omega \times 0.9 = 0.171\ \Omega$ and $0.19\ \Omega \times 1.9 = 0.361\ \Omega$.

TABLE 6.2 **The Zeta converter parameters (see Fig. 6.2).**

	Nominal Value	Variations
Input DC source voltage, Vg	20 V	±20%
MOSFET Drain-Source resistance, rds	10 mΩ	±20%
Capacitor, C_1	100 μF	±20%
Capacitor C_1 Equivalent Series Resistance (ESR), rC_1	0.19 Ω	[−10%,+90%]
Capacitor, C_2	220 μF	±20%
Capacitor C_2 Equivalent Series Resistance (ESR), rC_1	0.095 Ω	[−10%,+90%]
Inductor, L_1	100 μH	±10%
Inductor ESR, rL_1	1 mΩ	[−10%,+90%]
Inductor, L_2	55 μH	±10%
Inductor ESR, rL_2	0.55 mΩ	[−10%,+90%]
Diode voltage drop, vD	0.7 V	±30%
Diode forward resistance, rD	10 mΩ	[−10%,+50%]
Load resistor, R	6 Ω	±80%
Switching Frequency, Fsw	100 kHz	–

The schematic shown in Fig. 6.28 can be used to extract the additive uncertainty model of the studied Zeta converter. This schematic is used to extract the uncertain model of $\frac{v_o(s)}{d(s)}$ transfer function.

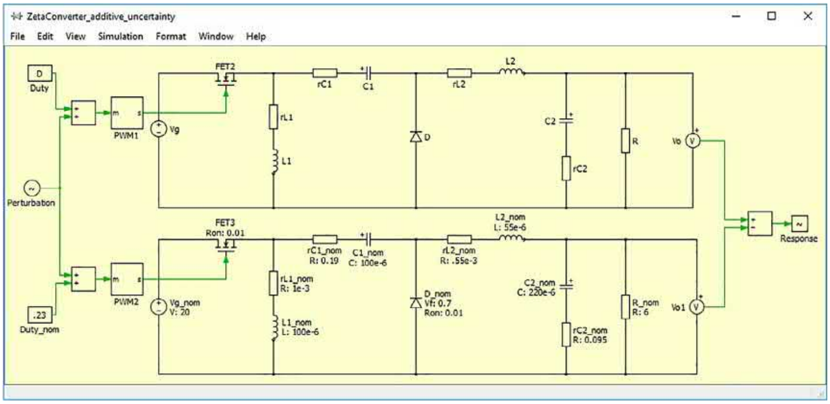
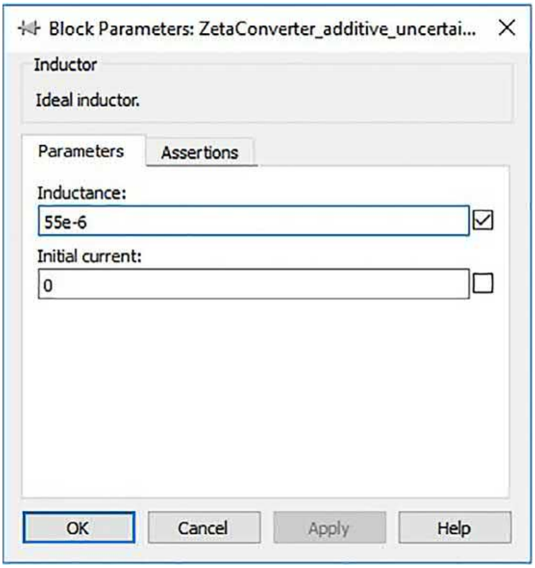

ZetaConverter_additive_uncertainty
File Edit View Simulation Format Window Help
D
Duty
Perturbation
.23
Duty_nom
PWM1
PWM2
Vg
Vg_nom
V: 20
FET2
FET3
Ron: 0.01
rC1
C1
rL1
L1
D
rL2
L2
C2
rC2
R
Vo
rC1_nom
R: 0.19
C1_nom
C: 100e-6
rL1_nom
R: 1e-3
L1_nom
L: 100e-6
D_nom
Vf: 0.7
Ron: 0.01
rL2_nom
R: .55e-3
L2_nom
L: 55e-6
C2_nom
C: 220e-6
rC2_nom
R: 0.095
R_nom
R: 6
Vo1
Response

FIG. 6.28 Schematic to extract the additive uncertainty model of the studied Zeta converter.

The schematic is composed of two Zeta converters. The lower Zeta converter has the nominal components. Each component of lower converter is double clicked and the nominal value of component is entered. For instance, value of the inductor connected to the output capacitor is set to 55 μH (Fig. 6.29).


Block Parameters: ZetaConverter_additive_uncertai...

Inductor

Ideal inductor.

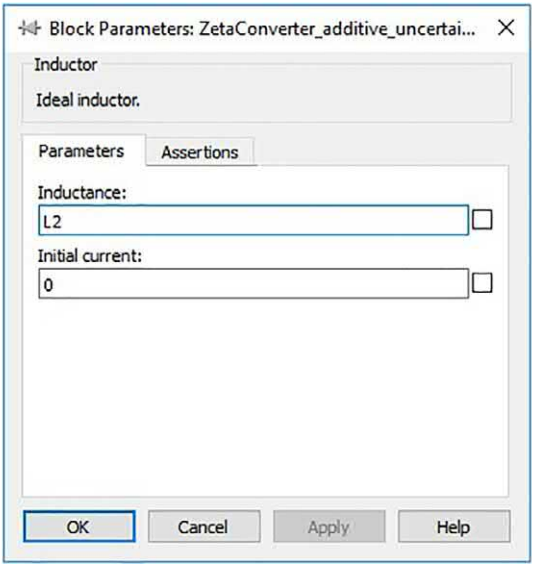
Inductance: 55e-6 ☑

Initial current: 0 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 6.29 The “Inductance” box is filed with a numeric value.

The upper Zeta converter will be supplied with the components selected randomly from their allowed interval (A MATLAB® program will produce the random values). In this section of schematic, value of each component is controlled with a variable, which will be initialized in the beginning of simulation. For instance, value of the inductor connected to the output capacitor is set by a variable named L2 (Fig. 6.30). According to Table 6.2, $L_{2,min} < L_2 < L_{2,max}$, where $L_{2,min} = 0.55\,\mu\text{H} \times 0.9 = 0.495\,\mu\text{H}$ and $L_{2,max} = 0.55\,\mu\text{H} \times 1.1 = 0.605\,\mu\text{H}$.


Block Parameters: ZetaConverter_additive_uncertai...

Inductor

Ideal inductor.

Inductance: L2

Initial current: 0

FIG. 6.30 The “Inductance” box is filed with a variable name.

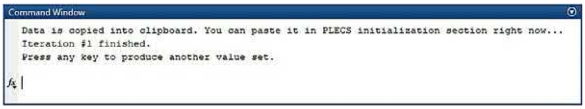
The following code produces acceptable component values (according to the variations given in Table 6.2) for the upper Zeta converter. The produced components set is copied into the Windows® clipboard, so one can paste it easily in the “Initialization” section of “Simulation Parameters” window and run the simulation with the generated random values.

```
%This program produces random component values
%produced values are copied into the Windows clipboard
%so you can paste them easily into the Initialization
%section of PLECS
clc
clear all


NumberOfIteration=20;   % set the desired number of iteration here.
DesiredOutputVoltage=5; % set the desired output voltage here.


n=0;
for i=1:NumberOfIteration
n=n+1;
%Definition of uncertainity in parameters
VG_unc=ureal('VG_unc',20,'Percentage',[-20 +20]);    % Value of input DC source is in the
range of 16..24
rg=0;                                                %  Internal  resistance  of  input  DC
source
```

```
rds_unc=ureal('rds_unc',.01,'Percentage',[-20 +20]);     % MOSFET on resistance
C1_unc=ureal('C1_unc',100e-6,'Percentage',[-20 +20]);    % Capacitor C1 value
C2_unc=ureal('C2_unc',220e-6,'Percentage',[-20 +20]);    % Capacitor C2 value
rC1_unc=ureal('rC1_unc',.19,'Percentage',[-10 +90]);     % Capacitor C1
Equivalent Series Resistance(ESR)
rC2_unc=ureal('rC2_unc',.095,'Percentage',[-10 +90]);    % Capacitor C2
Equivalent Series Resistance(ESR)
L1_unc=ureal('L1_unc',100e-6,'Percentage',[-10 +10]);    % Inductor L1 value
L2_unc=ureal('L2_unc',55e-6,'Percentage',[-10 +10]);     % Inductor L2 value
rL1_unc=ureal('rL1_unc',1e-3,'Percentage',[-10 +90]);    % Inductor L1
Equivalent Series Resistance(ESR)
rL2_unc=ureal('rL2_unc',.55e-3,'Percentage',[-10 +90]);  % Inductor L2
Equivalent Series Resistance(ESR)
rD_unc=ureal('rD_unc',.01,'Percentage',[-10 +50]);       % Diode series resistance
VD_unc=ureal('VD_unc',.7,'Percentage',[-30 +30]);        % Diode voltage drop
R_unc=ureal('R_unc',6,'Percentage',[-80 +80]);           % Load resistance
IO=0;                                                    % Average value of output current
source
fsw=100e3;                                               % Switching frequency

%Sampling the uncertain set
%for instance usample(VG_unc,1) takes one sample of uncertain
%parameter VG_unc

VG=usample(VG_unc,1);                                    % Sampled value of input
DC source
rds=usample(rds_unc,1);                                  % Sampled MOSFET on resistance
C1=usample(C1_unc,1);                                    % Sampled capacitor C1 value
C2=usample(C2_unc,1);                                    % Sampled capacitor C2 value
rC1=usample(rC1_unc,1);                                  % Sampled capacitor C1
Equivalent Series Resistance(ESR)
rC2=usample(rC2_unc,1);                                  % Sampled capacitor C2
Equivalent Series Resistance(ESR)
L1=usample(L1_unc,1);                                    % Sampled inductor L1 value
L2=usample(L2_unc,1);                                    % Sampled inductor L2 value
rL1=usample(rL1_unc,1);                                  % Sampled inductor L1
Equivalent Series Resistance(ESR)
rL2=usample(rL2_unc,1);                                   % Sampled inductor L2
Equivalent Series Resistance(ESR)
rD=usample(rD_unc,1);                                    % Sampled diode series
resistance
VD=usample(VD_unc,1);                                    % Sampled diode voltage drop
R=usample(R_unc,1);                                      % Sampled load resistance

%output voltage of an IDEAL(i.e. no losses) Zeta converter operating in CCM is given by:
%  D
%VO=--------VG
% 1-D
%where
%VO: average value of output voltage
%D: Duty Ratio
%VG: Input DC voltage
%So, for a IDEAL converter
% VO
%D=----------
% VO+VG
%Since our converter has losses we use a bigger duty ratio, for instance:
%   VO
%D=1.1 ----------
%  VO+VG
D=1.1*DesiredOutputVoltage/(VG+DesiredOutputVoltage);

%preparing the strings
S1=strcat('VG=',num2str(VG),';');
S2=strcat('rds=',num2str(rds),';');
```

```
S3=strcat('C1=',num2str(C1),';');
S4=strcat('C2=',num2str(C2),';');
S5=strcat('rC1=',num2str(rC1),';');
S6=strcat('rC2=',num2str(rC2),';');
S7=strcat('L1=',num2str(L1),';');
S8=strcat('L2=',num2str(L2),';');
S9=strcat('rL1=',num2str(rL1),';');
S10=strcat('rL2=',num2str(rL2),';');
S11=strcat('rD=',num2str(rD),';');
S12=strcat('VD=',num2str(VD),';');
S13=strcat('R=',num2str(R),';');
S14=strcat('D=',num2str(D),';');

%coping the data into the Windows Clipboard. So, you can paste it into the
%PLECS
data=strcat(S1,S2,S3,S4,S5,S6,S7,S8,S9,S10,S11,S12,S13,S14);
clipboard('copy',data)
disp('Data is copied into clipboard. You can paste it in PLECS initialization section right
now...')
message=strcat('Iteration #',num2str(n),' finished.');
disp(message)
disp('Press any key to produce another value set.')
disp(' ')
pause
end
disp('--------------------------')
disp('Program terminates here...')
```

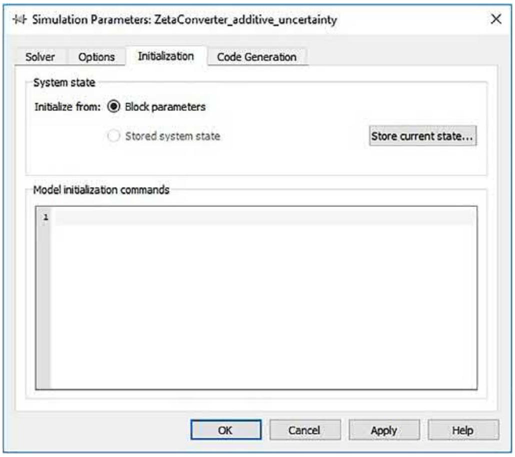
After running the code (and finishing the first iteration), the message shown in Fig. 6.31 will appear.


Command Window

```
Data is copied into clipboard. You can paste it in PLECS initialization section right now...
Iteration #1 finished.
Press any key to produce another value set.
```

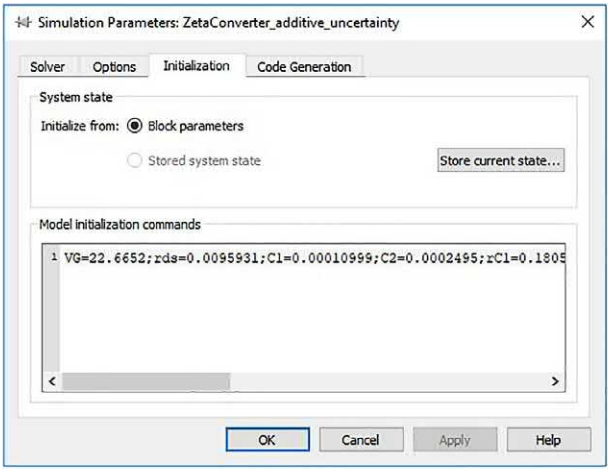
FIG. 6.31 User is asked to press a key before the next iteration.

Before pressing any key in the MATLAB® environment and starting the next iteration (which generates new random values), the user must go to the "Initialization" tab of "Simulation Parameters" window and paste the produced data (Figs. 6.32 and 6.33).


Simulation Parameters: ZetaConverter_additive_uncertainty
Solver
Options
Initialization
Code Generation
System state
Store current state...
Model initialization commands
1
OK
Cancel
Apply
Help

Initialize from: ● Block parameters
○ Stored system state

FIG. 6.32 The "Initialization" tab of "Simulation Parameters" window.


Simulation Parameters: ZetaConverter_additive_uncertainty
Solver
Options
Initialization
Code Generation
System state
Store current state...
Model initialization commands
OK
Cancel
Apply
Help

Initialize from: ● Block parameters

○ Stored system state

```
VG=22.6652;rds=0.0095931;C1=0.00010999;C2=0.0002495;rC1=0.1805
```

FIG. 6.33 Produced random values are paste into the “Model initialization commands” section of “Initialization” tab. Click the Model initialization commands section of and press the Ctrl+V to paste the generated values.

Now the user runs the PLECS® simulation with the values produced by the MATLAB® code (Fig. 6.34).

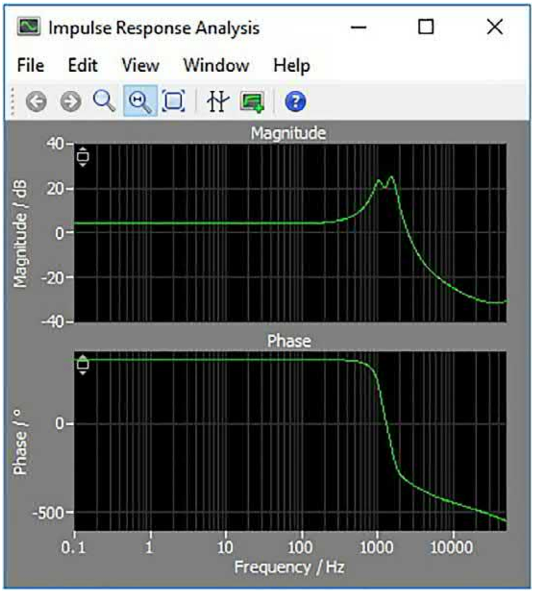


FIG. 6.34 Simulation result.

The user exports the obtained result as an .csv file (Fig. 6.35). The result is saved under the name of sim1.csv (Fig. 6.36).

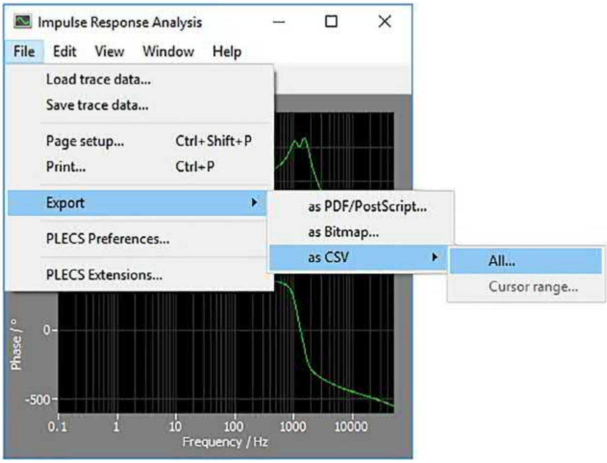
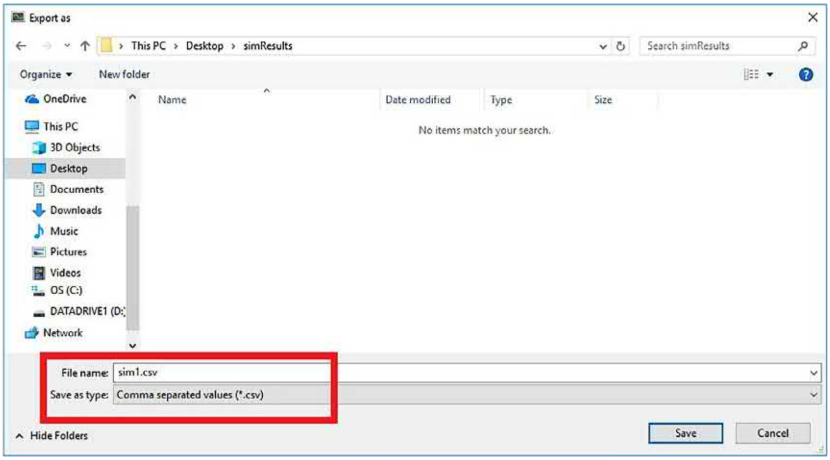

Impulse Response Analysis
File
Edit
View
Window
Help
Load trace data...
Save trace data...
Page setup... Ctrl+Shift+P
Print... Ctrl+P
Export
PLECS Preferences...
PLECS Extensions...
as PDF/PostScript...
as Bitmap...
as CSV
All...
Cursor range...
Phase / °
0
-500
0.1
1
10
100
1000
10000
Frequency / Hz

FIG. 6.35 Obtained result is exported as a .csv file.


Export as
This PC › Desktop › simResults
Search simResults
Organize
New folder
OneDrive
This PC
3D Objects
Desktop
Documents
Downloads
Music
Pictures
Videos
OS (C:)
DATADRIVE1 (D:)
Network
Name
Date modified
Type
Size
No items match your search.
Hide Folders
Save
Cancel

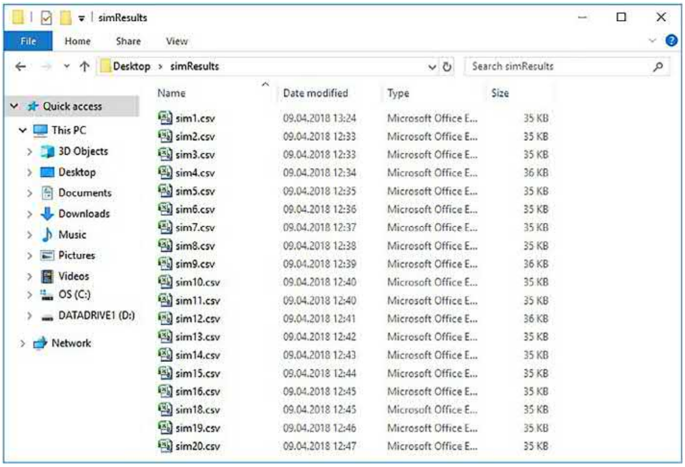
File name: sim1.csv

Save as type: Comma separated values (*.csv)

FIG. 6.36 The simulation result for the first generated random set is saved under the name of sim1.csv.

After saving the result, the user returns to the MATLAB® environment and press any key in order to produce a new random set. After the random set is generated, he/she switches to the PLECS® environment and pastes the produced random set into the Initialization tab of Simulation Parameters window.

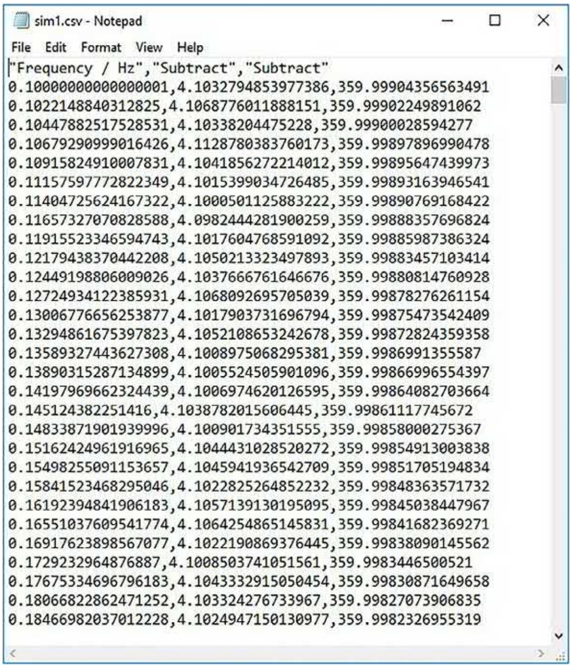
The user redoes the aforementioned process until all the iterations are finished. The number of iterations is set by the `NumberOfIteration` variable defined in the seventh line of the code (Fig. 6.37). `NumberOfIteration` is set to 20 in the shown code. So, the MATLAB® code produces 20 different random sets. All the simulation results are exported as .csv file.



Name	Date modified	Type	Size
sim1.csv	09.04.2018 13:24	Microsoft Office E...	35 KB
sim2.csv	09.04.2018 12:33	Microsoft Office E...	35 KB
sim3.csv	09.04.2018 12:33	Microsoft Office E...	35 KB
sim4.csv	09.04.2018 12:34	Microsoft Office E...	36 KB
sim5.csv	09.04.2018 12:35	Microsoft Office E...	35 KB
sim6.csv	09.04.2018 12:36	Microsoft Office E...	35 KB
sim7.csv	09.04.2018 12:37	Microsoft Office E...	35 KB
sim8.csv	09.04.2018 12:38	Microsoft Office E...	35 KB
sim9.csv	09.04.2018 12:39	Microsoft Office E...	36 KB
sim10.csv	09.04.2018 12:40	Microsoft Office E...	35 KB
sim11.csv	09.04.2018 12:40	Microsoft Office E...	35 KB
sim12.csv	09.04.2018 12:41	Microsoft Office E...	36 KB
sim13.csv	09.04.2018 12:42	Microsoft Office E...	35 KB
sim14.csv	09.04.2018 12:43	Microsoft Office E...	35 KB
sim15.csv	09.04.2018 12:44	Microsoft Office E...	35 KB
sim16.csv	09.04.2018 12:45	Microsoft Office E...	35 KB
sim18.csv	09.04.2018 12:45	Microsoft Office E...	35 KB
sim19.csv	09.04.2018 12:46	Microsoft Office E...	35 KB
sim20.csv	09.04.2018 12:47	Microsoft Office E...	35 KB

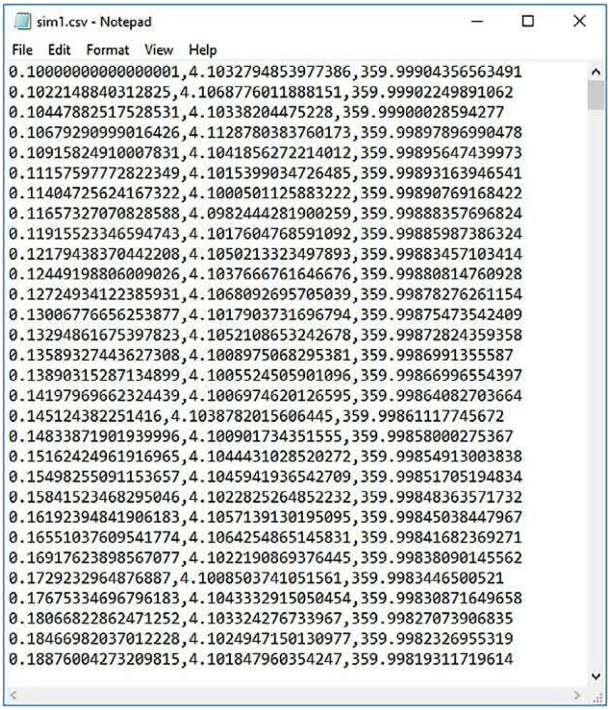
FIG. 6.37 Analysis results are saved as sim1.csv,sim2.csv, sim3.csv, …, sim20.csv.

All the obtained .csv files are opened in the Notepad and their first line is removed (Figs. 6.38 and 6.39).



```
"Frequency / Hz","Subtract","Subtract"
0.10000000000000001,4.1032794853977386,359.99904356563491
0.1022148840312825,4.1068776011888151,359.99902249891062
0.10447882517528531,4.10338204475228,359.99900028594277
0.10679290999016426,4.1128780383760173,359.99897896990478
0.10915824910007831,4.1041856272214012,359.99895647439973
0.11157597772822349,4.1015399034726485,359.99893163946541
0.11404725624167322,4.1000501125883222,359.99890769168422
0.11657327070828588,4.0982444281900259,359.99888357696824
0.11915523346594743,4.1017604768591092,359.99885987386324
0.12179438370442208,4.1050213323497893,359.99883457103414
0.12449198806009026,4.1037666761646676,359.99880814760928
0.12724934122385931,4.1068092695705039,359.99878276261154
0.13006776656253877,4.1017903731696794,359.99875473542409
0.13294861675397823,4.1052108653242678,359.99872824359358
0.13589327443627308,4.1008975068295381,359.9986991355587
0.13890315287134899,4.1005524505901096,359.99866996554397
0.14197969662324439,4.1006974620126595,359.99864082703664
0.145124382251416,4.1038782015606445,359.99861117745672
0.14833871901939996,4.100901734351555,359.99858000275367
0.15162424961916965,4.1044431028520272,359.99854913003838
0.15498255091153657,4.1045941936542709,359.99851705194834
0.15841523468295046,4.1022825264852232,359.99848363571732
0.16192394841906183,4.1057139130195095,359.99845038447967
0.16551037609541774,4.1064254865145831,359.99841682369271
0.16917623898567077,4.1022190869376445,359.99838090145562
0.1729232964876887,4.1008503741051561,359.9983446500521
0.17675334696796183,4.1043332915050454,359.99830871649658
0.18066822862471252,4.103324276733967,359.99827073906835
0.18466982037012228,4.1024947150130977,359.9982326955319
```

FIG. 6.38 Opening the "sim1.csv" in Notepad.



```
0.10000000000000001,4.1032794853977386,359.99904356563491
0.1022148840312825,4.1068776011888151,359.99902249891062
0.10447882517528531,4.10338204475228,359.99900028594277
0.10679290999016426,4.1128780383760173,359.99897896990478
0.10915824910007831,4.1041856272214012,359.99895647439973
0.11157597772822349,4.1015399034726485,359.99893163946541
0.11404725624167322,4.1000501125883222,359.99890769168422
0.11657327070828588,4.0982444281900259,359.99888357696824
0.11915523346594743,4.1017604768591092,359.99885987386324
0.12179438370442208,4.1050213323497893,359.99883457103414
0.12449198806009026,4.1037666761646676,359.99880814760928
0.12724934122385931,4.1068092695705039,359.99878276261154
0.13006776656253877,4.1017903731696794,359.99875473542409
0.13294861675397823,4.1052108653242678,359.99872824359358
0.13589327443627308,4.1008975068295381,359.9986991355587
0.13890315287134899,4.1005524505901096,359.99866996554397
0.14197969662324439,4.1006974620126595,359.99864082703664
0.145124382251416,4.1038782015606445,359.99861117745672
0.14833871901939996,4.100901734351555,359.99858000275367
0.15162424961916965,4.1044431028520272,359.99854913003838
0.15498255091153657,4.1045941936542709,359.99851705194834
0.15841523468295046,4.1022825264852232,359.99848363571732
0.16192394841906183,4.1057139130195095,359.99845038447967
0.16551037609541774,4.1064254865145831,359.99841682369271
0.16917623898567077,4.1022190869376445,359.99838090145562
0.1729232964876887,4.1008503741051561,359.9983446500521
0.17675334696796183,4.1043332915050454,359.99830871649658
0.18066822862471252,4.103324276733967,359.99827073906835
0.18466982037012228,4.1024947150130977,359.9982326955319
0.18876004273209815,4.101847960354247,359.99819311719614
```

FIG. 6.39 First line of sim1.csv is removed.

Following MATLAB® code reads all the simulation results (prepared .csv files) and draws them simultaneously on the same graph.

```
%This program draws the results(Additive Uncertainty) produced by PLECS
clc
clear all
NumberOfSimulations=20;


for i=1:NumberOfSimulations
disp('percentage of work done:')
disp(i/NumberOfSimulations*100)
disp(' ')


name=strcat('sim',num2str(i),'.csv');
x=csvread(strcat('C:\Users\Desktop\SimResults\',name)); %reads the csv file
f=x(:,1);
w=2*pi*f;
M=10.^(x(:,2)/20).*exp(j*x(:,3)*pi/180); %Magnitude of freq. resp.(complex form)
H=frd(M/Mnominal,w);
bode(H), grid on
hold on


end
title('Additive uncertainty in vo(s)/d(s)')
```

After running the code, the result shown in Fig. 2.107 is obtained.

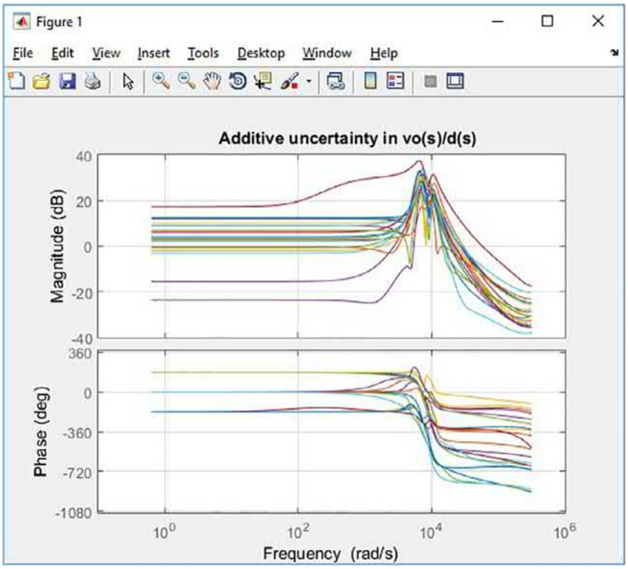


FIG. 6.40 Additive uncertainty in $\frac{v_o(s)}{d(s)}$.

After the graph shown in Fig. 6.40 is obtained, we need to fit such a weight, which passes above all the transfer functions. The following code is used for this purpose.

```
%Selection of the upper bound for uncertainty
Number_of_points=20;
[freq,resp_dB]=ginput(Number_of_points);
%Reading are in decibel(dB). The following loop find the magnitudes.
for i=1:Number_of_points
    resp(i)=10^(resp_dB(i)/20);
end
selected_points_frd=frd(resp,freq); %Making the frd object.
ord=3; %Order of produced weight
W=fitmagfrd(selected_points_frd,ord); %Fitting a transfer function to the selected data
points.
Wtf=tf(W);
bode(Wtf,'r--')
```

After the code is run, the user clicks 20 points, which are above the highest transfer function. For instance, some of the acceptable points are shown with red stars in Fig. 6.41.

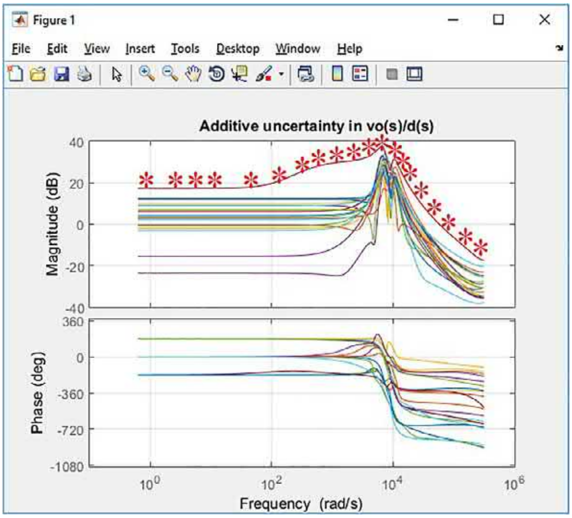


Figure 1
File Edit View Insert Tools Desktop Window Help
Additive uncertainty in vo(s)/d(s)
Magnitude (dB)
40
20
0
-20
-40
Phase (deg)
360
0
-360
-720
-1080
10^0
10^2
10^4
10^6
Frequency (rad/s)

FIG. 6.41 In order to estimate the uncertainty weight, the user selects points, which are above the highest transfer function. Use points of magnitude plot.

After selection of 20 points is finished, the `fitmagfrd` command estimates a transfer function for the selected points. Order of estimated transfer function is defined by the `ord` variable.

6.4.4.2 *Multiplicative uncertainty model*

Schematic shown in Fig. 6.28 can be used to extract the additive uncertainty model of the converter. Following code normalizes (divide the difference to the nominal transfer function) the results of previous analysis. This lets the calculation of multiplicative uncertainty weights.

```
%This program draws the results(Multiplicative Uncertainty) produced by PLECS
%Running this code takes time, please be patient!
clc
clear all
NumberOfSimulations=20;


%Results obtained for nominal system
%see previous analyses
x=csvread(strcat('C:\Users\Dekanlik03\Desktop\nominal_values_freq_resp.csv'));
f=x(:,1);
w=2*pi*f;
M_nominal=10.^(x(:,2)/20).*exp(j*x(:,3)*pi/180); %Magnitude of freq. resp.(complex form)
vo_d_nominal=frd(M_nominal,w);


for i=1:NumberOfSimulations
disp('percentage of work done:')
disp(i/NumberOfSimulations*100)
disp(' ')


name=strcat('sim',num2str(i),'.csv');
x=csvread(strcat('C:\Users\Dekanlik03\Desktop\SimResults\',name)); %reads the csv file
f=x(:,1);
```

```
w=2*pi*f;
M=10.^(x(:,2)/20).*exp(j*x(:,3)*pi/180); %frequency response complex form for the read
.csv file
H=frd(M/M_nominal,w);
bode(H), grid on
hold on
end
title('Multiplicative uncertainty in plant')
```

After running the code, the result shown in Fig. 6.42 is obtained.

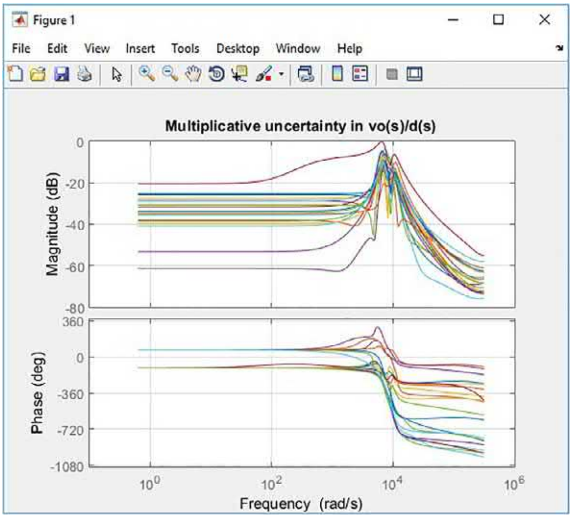


Figure 1
File Edit View Insert Tools Desktop Window Help
Multiplicative uncertainty in vo(s)/d(s)
Magnitude (dB)
0
-20
-40
-60
-80
Phase (deg)
360
0
-360
-720
-1080
10^0
10^2
10^4
10^6
Frequency (rad/s)

FIG. 6.42 Multiplicative uncertainty in $\frac{v_o(s)}{d(s)}$.

An upper bound for the transfer functions shown in Fig. 6.42 can be found with the aid of following code. Obtained weight is shown in Fig. 6.43.

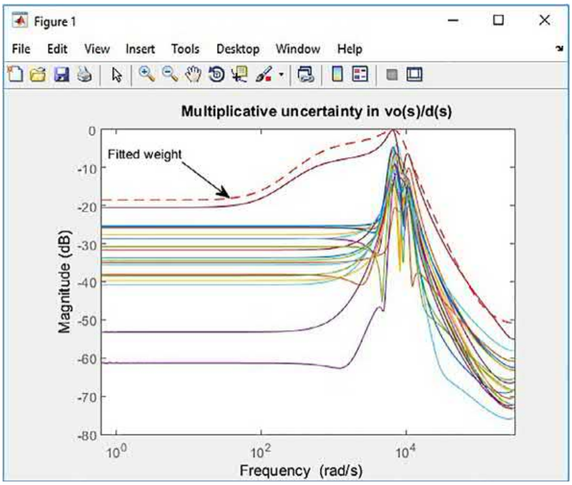
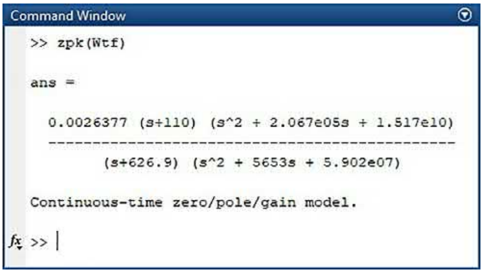


Figure 1
File Edit View Insert Tools Desktop Window Help
Multiplicative uncertainty in vo(s)/d(s)
Fitted weight
Magnitude (dB)
0
-10
-20
-30
-40
-50
-60
-70
-80
10^0
10^2
10^4
Frequency (rad/s)

FIG. 6.43 Fitting a multiplicative uncertainty weight.

```
%This program draws the results(Multiplicative Uncertainty) produced by PLECS
%You can obtain the upper bound of uncertainty as well.
%Running this code takes time, please be patient!
clc
clear all
NumberOfSimulations=20;


%Results obtained for nominal system
%see previous analyses
x=csvread(strcat('C:\Users\Desktop\nominal_values_freq_resp.csv'));
f=x(:,1);
w=2*pi*f;
M_nominal=10.^(x(:,2)/20).*exp(j*x(:,3)*pi/180); %Magnitude of freq. resp.(complex form)
vo_d_nominal=frd(M_nominal,w);


for i=1:NumberOfSimulations
disp('percentage of work done:')
disp(i/NumberOfSimulations*100)
disp(' ')


name=strcat('sim',num2str(i),'.csv');
x=csvread(strcat('C:\Users\Desktop\SimResults\',name)); %reads the csv file
f=x(:,1);
w=2*pi*f;
M=10.^(x(:,2)/20).*exp(j*x(:,3)*pi/180); %frequency response complex form
H=frd(M/M_nominal,w);
bodemag(H), grid on
hold on
end
title('Multiplicative uncertainty in plant')


pause
disp('press any key to continue')


%Selection of the upper bound for uncertainty
Number_of_points=20;
[freq,resp_dB]=ginput(Number_of_points);
%Reading are in desibel(dB). The following loop find the magnitudes.
for i=1:Number_of_points
    resp(i)=10^(resp_dB(i)/20);
end
selected_points_frd=frd(resp,freq); %Making the frd object.
ord=3; %Order of produced weight
W=fitmagfrd(selected_points_frd,ord); %Fitting a transfer function to the selected data
points.
Wtf=tf(W);
bode(Wtf,'r--')
```

One can obtain the calculated weight with the aid of commands shown in Fig. 6.44.



```
>> zpk(Wtf)

ans =

  0.0026377 (s+110) (s^2 + 2.067e05s + 1.517e10)
  ----------------------------------------------
        (s+626.9) (s^2 + 5653s + 5.902e07)

Continuous-time zero/pole/gain model.

>> 
```

FIG. 6.44 Equation of obtained weight.

References

[1] D. Gu, P. Petkov, M. Konstantinov, Robust control design with MATLAB, second ed, Springer-Verlag, London, 2013.

[2] R. Barmish, New Tools for Robustness of Linear Systems, first ed, Macmillan, USA, 1993.

[3] S. Bhattacharyya, H. Chapellat, L. Keel, Robust Control the Parametric Approach, USA, Prentice Hall PTR, 1995.

[4] P. Lundstron, S. Skogestad, Z. Wang, Performance weight selection for H_∞ and μ control method, Trans. Instrum. Control 31 (1991) 241–252.

[5] S. Skogesttad, I. Postlethwaite, Multivariable Feedback Control—Analysis and Design, second ed, John Wiley & Sons, USA, 2005.

[6] R. Beaven, M. Wright, D. Seaward, Weighting function selection in the H_∞ design process, Control Eng. Pract. (1996) 625–633.

[7] D. Donha, M. Katebi, Automatic weight selection for H_∞ controller synthesis, Int. J. Syst. Sci. (2007) 651–664.

[8] E. Alfaro-Cid, E. McGookin, D. Murray-Smith, Optimization of the weighting functions of an H_∞ controller using genetic algorithms and structured genetic algorithms, Int. J. Syst. Sci. (2008) 335–347.

[9] G. Zames, Feedback and optimal sensitivity: model reference transformations, multiplicative semi norms, and approximate inverses, IEEE Trans. Automat. Control (1981) 301–320.

[10] H. Kwakernak, Robust control and H_∞ optimization, Automatica (1993) 255–273.

[11] K. Zhou, J. Doyle, Essential of Robust Control, Pearson, USA, 1997.

[12] M. Green, D. Limbeer, Linear Robust Control, USA, Dover Publications, 2012.

[13] R. Chiang, M. Safonov, G. Balas, et al., Robust Control Toolbox, third ed., USA, The MathWorks Inc, 2007.

[14] F. Asadi, S. Pongswatd, et al., Modeling Uncertainties in DC-DC Converters with MATLAB and PLECS, Morgan & Claypool, USA, 2018.

Chapter 7

Simulation of magnetic circuits in PLECS

Chapter outline

7.1 Introduction 399
7.2 Magnetic blocks 399
  7.2.1 Winding block 400
  7.2.2 Magnetic permeance block 402
  7.2.3 Saturable core block 402
  7.2.4 Hysteretic core block 404
  7.2.5 Air gap block 405
  7.2.6 Leakage flux path block 405
7.3 Implementation of blocks 405
7.4 Some commonly used magnetic configuration 406
7.5 Case study 407
7.6 Where to go next? 417
Reference 419
Further reading 419

7.1 Introduction

Magnetic components play an important role in power electronics converters. Using the powerful ready-to-use blocks in the Magnetic section of Library Browser, one can simulate complex magnetic circuits easily. Simulation of nonlinear effects such as hysteresis, saturation, and flux leakage is quite easy with the aid of blocks inside the Magnetic section of Library Browser.

This chapter focuses on the blocks inside the Magnetic section of Library Browser. It is assumed that the reader knows the basics of magnetic circuits. Chapter 1 of [1] is a good reference if you need to review the concepts. It is highly recommended to read the provided help for blocks inside the magnetic library of PLECS in order to learn the details of blocks.

This is the last chapter of book, which educates the software details for simulation of power electronics converters using PLECS. However, the world of simulation of power electronics converters using PLECS is not limited at all. Keeping this in mind, the last part of this chapter shows how you can increase your knowledge of simulation. It doesn't give you any fish, it teaches you fishing!

7.2 Magnetic blocks

The Magnetic section of Library Browser is shown in Fig. 7.1.

Simulation of Power Electronics Converters Using PLECS®. https://doi.org/10.1016/B978-0-12-817364-0.00007-2
© 2020 Elsevier Inc. All rights reserved.

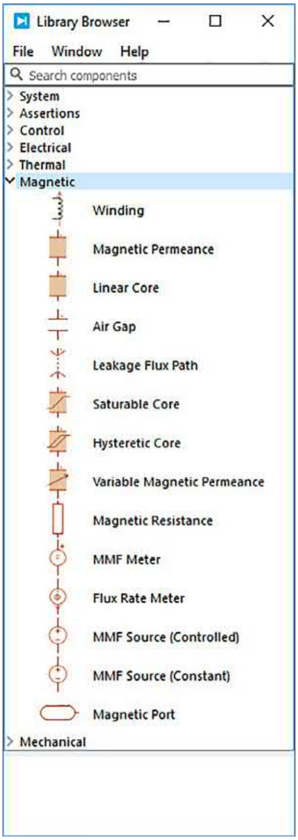

Library Browser
File
Window
Help
Search components
System
Assertions
Control
Electrical
Thermal
Magnetic
Winding
Magnetic Permeance
Linear Core
Air Gap
Leakage Flux Path
Saturable Core
Hysteretic Core
Variable Magnetic Permeance
Magnetic Resistance
MMF Meter
Flux Rate Meter
MMF Source (Controlled)
MMF Source (Constant)
Magnetic Port
Mechanical

FIG. 7.1 The Magnetic Section of Library Browser.

The blocks inside the Magnetic section of Library Browser can be combined with components from other domains in order to do the simulation. For instance, notice the simulation shown in Fig. 7.2. The blocks inside the rectangle simulate the flyback transformer. This simulation contains electrical blocks, control blocks, and magnetic blocks.

Some of the important Magnetic blocks are studied here.

7.2.1 Winding block

The Winding block is the interface between the electrical and magnetic domains. In order to specify the number of turns, double click the Winding block and fill the Number of turns box (Fig. 7.3). It is quite useful to click the Help button and study the provided information about the block. Note that only the Winding block (see Fig. 7.1) can be connected to components from the power circuit. Other blocks of Magnetic library can't be connected to power circuit.

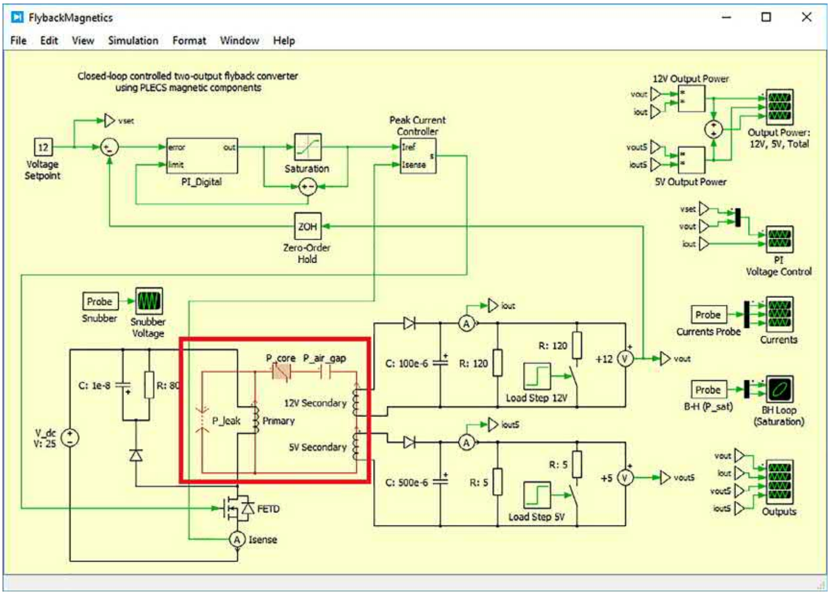
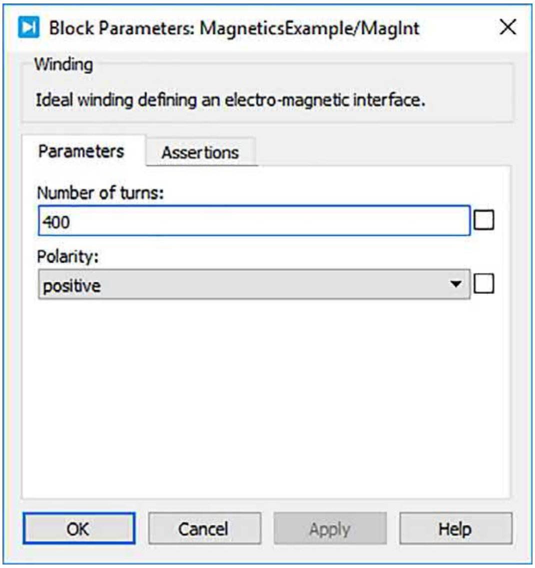

FlybackMagnetics
File Edit View Simulation Format Window Help
Closed-loop controlled two-output flyback converter
using PLECS magnetic components
vset
12
Voltage Setpoint
error
limit
out
PI_Digital
Saturation
Peak Current Controller
Iref
Isense
ZOH
Zero-Order Hold
12V Output Power
vout
iout
vout5
iout5
5V Output Power
Output Power: 12V, 5V, Total
vset
vout
iout
PI Voltage Control
Probe
Snubber
Snubber Voltage
C: 1e-8
R: 80
V_dc
V: 25
P_core
P_air_gap
P_leak
Primary
12V Secondary
5V Secondary
FETD
Isense
iout
C: 100e-6
R: 120
R: 120
Load Step 12V
+12
vout
iout5
C: 500e-6
R: 5
R: 5
Load Step 5V
+5
vout5
Probe
Currents Probe
Currents
Probe
B-H (P_sat)
BH Loop (Saturation)
vout
iout
vout5
iout5
Outputs

FIG. 7.2 An example of simulation, which uses the blocks from the Magnetic library. With the aid of blocks from the Magnetic library, more realistic simulations can be done.


Block Parameters: MagneticsExample/MagInt

Winding

Ideal winding defining an electro-magnetic interface.

Number of turns: 400

Polarity: positive

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 7.3 Parameters of Winding block.

If you want to simulate the winding copper resistance, you must attach a resistor block (selected from Passive Components section of Library Browser) to the electrical side (Fig. 7.4).

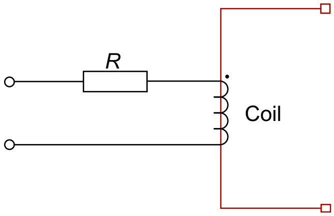


FIG. 7.4 Copper losses are modeled with a resistor added in series with the winding.

Eddy current losses are modeled by a resistance on the magnetic side.

7.2.2 Magnetic permeance block

The Magnetic Permeance block (see Fig. 7.1) establishes a linear relationship between the magnetic flux φ and the magnetomotive force $\mathcal{F}$. Magnetomotive permeance $\mathcal{P}$ is the reciprocal of magnetic reluctance $\mathcal{R}$:

$$\mathcal{P}=\frac{1}{\mathcal{R}}=\frac{\varphi}{\mathcal{F}}$$

The Linear Core block, models a segment of magnetic core. It establishes a linear relationship between the magnetic flux φ and the magnetomotive force $\mathcal{F}$

$$\frac{\varphi}{\mathcal{F}}=\frac{\mu_0\mu_r A}{l},$$

where $\mu_0=4\pi\times 10^{-7}\,\frac{N}{A^2}$ is the permeability of free space, μ_r is the relative permeability of the core material, A is the cross-section area, and l is the length of flux path.

7.2.3 Saturable core block

Typical $B-H$ curve for a ferrite core is shown in Fig. 7.5. After a certain point, though, further increases in the magnetomotive force produce relatively smaller increases in the flux. Finally, an increase in the magnetomotive force produces almost no change at all. The region of this Fig. in which the curve flattens out is called the saturation region, and the core is said to be saturated. In contrast, the region where the flux changes very rapidly is called the unsaturated region of the curve, and the core is said to be unsaturated. The transition region between the unsaturated region and the saturated region is sometimes called the knee of the curve.

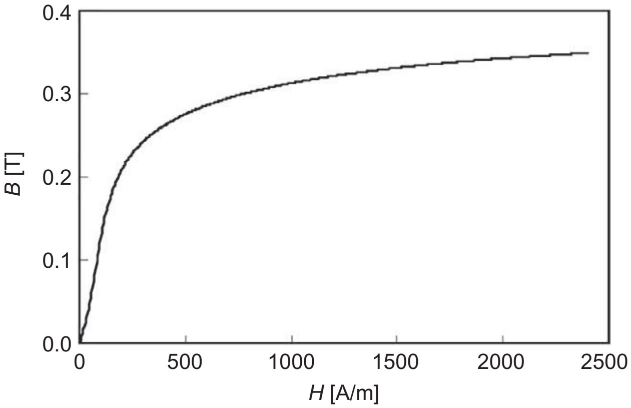


FIG. 7.5 Typical $B-H$ curve for a ferrite core.

Using the Saturable Core block, you can simulate the saturation. This block defines a nonlinear relationship between the field strength H and the flux density B to model the saturation. PLECS provides two fitting functions: atan fit and coth fit. The $B-H$ equation for arctangent fit has the following equation:

$$B = \frac{2}{\pi} B_{\text{sat}} \tan^{-1}\left(\frac{\pi H}{2a}\right) + \mu_{\text{sat}} H$$

And the $B-H$ equation for cotangent hyperbolic fit is:

$$B = B_{\text{sat}}\left(\coth\frac{3H}{a} - \frac{a}{3H}\right) + \mu_{\text{sat}} H$$

Both fitting functions have three degrees of freedom, which are set by coefficients μ_{sat}, B_{sat} and a. μ_{sat} is the fully saturated permeability, which usually corresponds to the magnetic constant μ_0, i.e., the permeability of air. B_{sat} defines the knee of the saturation transition between unsaturated and saturated permeability:

$$B_{\text{sat}} = (B - \mu_{\text{sat}} H)|_{H \to \infty}$$

The coefficient a is determind by the unsaturated permeability μ_{unsat} at $H=0$.

$$a = \frac{B_{\text{sat}}}{\mu_{\text{unsat}} - \mu_{\text{sat}}}$$

The typical $B-H$ curve, which is produced by these types of equations, is shown in Fig. 7.6.

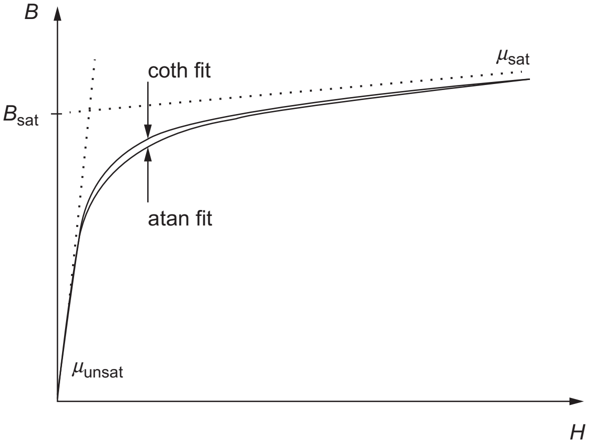


FIG. 7.6 Typical B–H curves, which are obtainable with the provided fitting functions.

7.2.4 Hysteretic core block

Simulation of hysteresis is possible with the aid of Hysteretic Core block. The relationship between the field strength H and the flux density B is nonlinear. The hysteresis characteristics are based on a Preisach model with a Lorentzian distribution function. Fig. 7.7 shows a fully excited major hysteresis curve with some minor reversal loops. The major curve is defined by the saturation point (H_{sat}, B_{sat}), the coercitive field strength H_c, the remanence flux density B_r, and the saturated permeability μ_{sat}.

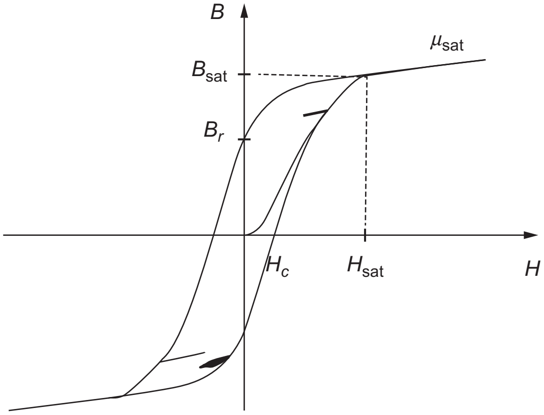


FIG. 7.7 Hysteresis curve with some minor reversal loops.

7.2.5 Air gap block

Air Gap block models an air gap in a magnetic core. The relation between the magnetic flux φ and magnetomotive force $\mathcal{F}$ for the block is

$$\frac{\varphi}{\mathcal{F}}=\frac{\mu_0 A}{l},$$

where $\mu_0=4\pi\times10^{-7}\frac{N}{A^2}$ is the permeability of free space, A is the cross-sectional area, and l is the length of the flux path.

7.2.6 Leakage flux path block

This component models a magnetic leakage flux path. Leakage Flux Path block is equivalent to the Magnetic Permeance block. The only difference is the symbol. The relation between magnetic flux φ and the magnetomotive force $\mathcal{F}$ is linear:

$$\mathcal{P}=\frac{1}{\mathcal{R}}=\frac{\varphi}{\mathcal{F}}$$

Magnetic permeance $\mathcal{P}$ is the reciprocal of magnetic reluctance $\mathcal{R}$.

7.3 Implementation of blocks

You can see the implementation details of blocks by right clicking the block and clicking the Look under mask (Fig. 7.8).

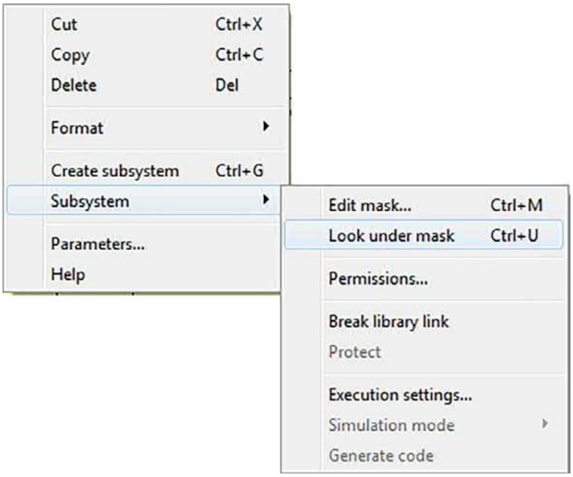
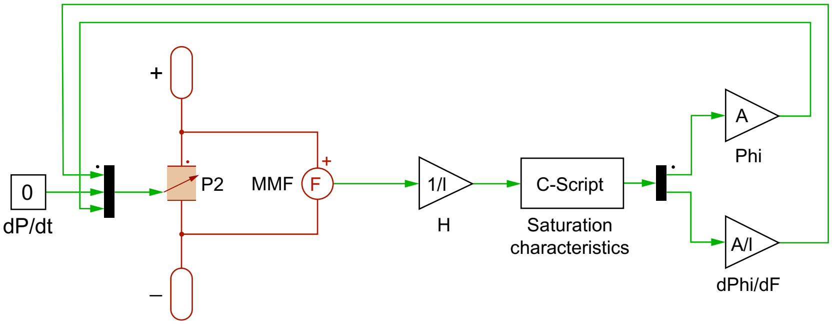

Cut Ctrl+X
Copy Ctrl+C
Delete Del
Format
Create subsystem Ctrl+G
Subsystem
Parameters...
Help
Edit mask... Ctrl+M
Look under mask Ctrl+U
Permissions...
Break library link
Protect
Execution settings...
Simulation mode
Generate code

FIG. 7.8 You can see the implementation of block with the aid of Look under mask.

For instance, if you right click on a Saturable Core block and click the Look under mask, the schematic shown in Fig. 7.9 will appear.

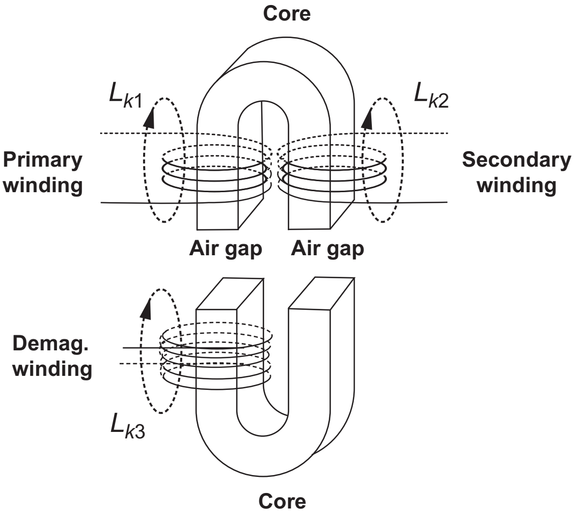


+
0
dP/dt
P2
MMF
F
+
−
1/l
H
C-Script
Saturation characteristics
A
Phi
A/l
dPhi/dF

FIG. 7.9 Implementation of Saturable Core block.

7.4 Some commonly used magnetic configuration

We study some of the commonly used configurations in this section. Assume a configuration such as the one shown in Fig. 7.10. This configuration can be modeled as shown in Fig. 7.11.



Core
L_{k1}
L_{k2}
Primary winding
Secondary winding
Air gap
Air gap
Demag. winding
L_{k3}
Core

FIG. 7.10 Mutually coupled inductors.

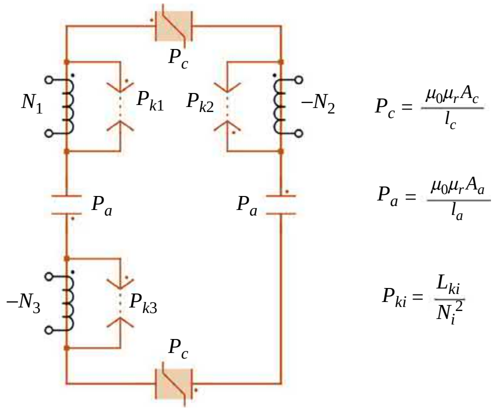

P_c
N_1
P_{k1}
P_{k2}
$-N_2$
$P_c = \frac{\mu_0\mu_r A_c}{l_c}$
P_a
P_a
$P_a = \frac{\mu_0\mu_r A_a}{l_a}$
$-N_3$
P_{k3}
$P_{ki} = \frac{L_{ki}}{N_i^2}$
P_c

FIG. 7.11 PLECS model of mutually coupled inductors shown in Fig. 7.10.

Figs. 7.12 and 7.13 give more examples on modeling the magnetic circuits.

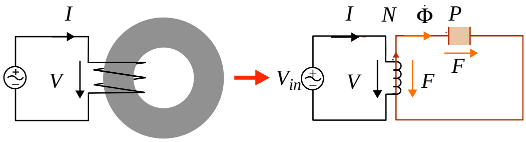

I
V
I
N
$\dot{\Phi}$
P
V_{in}
V
F
F

FIG. 7.12 One winding toroid core and its PLECS model.

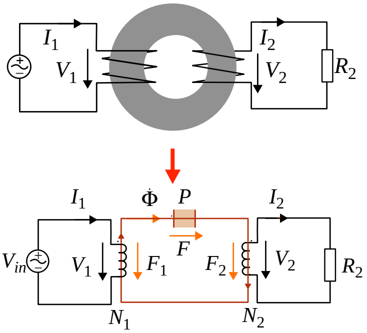

I_1
V_1
I_2
V_2
R_2
I_1
$\dot{\Phi}$
P
I_2
V_{in}
V_1
F_1
F
F_2
V_2
R_2
N_1
N_2

FIG. 7.13 Two winding toroid cores and its PLECS model.

7.5 Case study

Consider a configuration like the one shown in Fig. 7.14. This Fig. shows a ferromagnetic core whose mean path length is 40 cm. There is a small gap of 0.05 cm in the structure of the otherwise whole core. The cross-sectional area of the core is 12 cm^2, the relative permeability of the core is 4000, and the coil of wire on the core has 400 turns. Resistance of wire and input DC current are 10 Ω and 0.6 A, respectively. Fringing in the air gap is ignored. We want to calculate the flux density in the air gap.

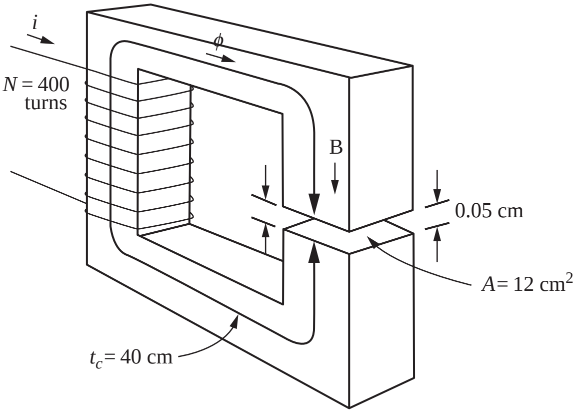


FIG. 7.14 400 Turns of wire wound a ferromagnetic core. The core has a small air gap with 0.05-cm length.

Fig. 7.15 shows the electrical equivalent circuit of magnetic circuit shown in Fig. 7.14.

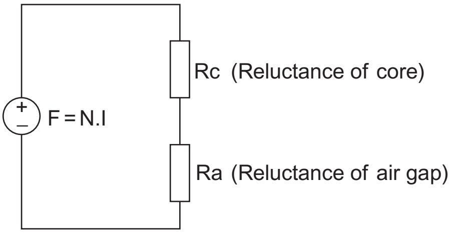


FIG. 7.15 The magnetic circuit corresponds to Fig. 7.14.

The reluctance of core is:

$$\mathcal{R}_c = \frac{l_c}{\mu_r \mu_0 A_c} = \frac{0.4\,\text{m}}{(4000) \times (4\pi \times 10^{-7}) \times 0.0012\,\text{m}^2} = 66298\ \text{A.turn/Wb}$$

The reluctance of air gap is:

$$\mathcal{R}_a = \frac{l_a}{\mu_r \mu_0 A_c} = \frac{0.0005\,\text{m}}{(4\pi \times 10^{-7}) \times 0.0012\,\text{m}^2} = 331490\ \text{A.turn/Wb}$$

The total reluctance is:

$$\mathcal{R}_t = \mathcal{R}_a + \mathcal{R}_c = 397788\ \text{A.turn/Wb}$$

The MMF distribution can be calculated easily:

$$\mathcal{F}_c = \frac{\mathcal{R}_c}{\mathcal{R}_t} = \frac{\mathcal{R}_c}{\mathcal{R}_a + \mathcal{R}_c} \times N \times I = \frac{66298}{397788} \times 400 \times 0.6 = 40\,\text{A.turn}$$

$$\mathcal{F}_a = \frac{\mathcal{R}_a}{\mathcal{R}_t} = \frac{\mathcal{R}_a}{\mathcal{R}_a + \mathcal{R}_c} \times N \times I = \frac{331490}{397788} \times 400 \times 0.6 = 200\,\text{A.turn}$$

The flux density of air gap is:

$$B = \frac{\varphi}{A} = \frac{\frac{N.I}{\mathcal{R}_t}}{A} = \frac{\frac{400 \times 0.6}{397788}}{0.0012} = 0.5\,\text{T}$$

The inductance can be calculated as:

$$L = \frac{N^2}{\mathcal{R}_t} = \frac{400^2}{397788} = 0.40\,\text{H}$$

In order to analyze the configuration shown in Fig. 7.14 with PLECS, open a new simulation file and add a Winding block to it (Fig. 7.16). After that, double click the Winding block and set the Number of turns box to 400.

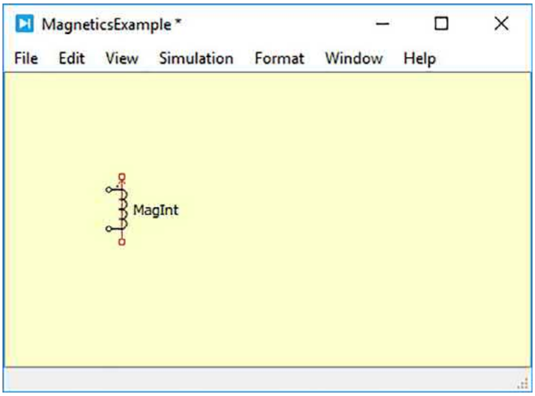


FIG. 7.16 Addition of a Winding block to the simulation file.

Add Linear Core and Air Gap blocks to the simulation file (Fig. 7.17). Do the settings as shown in Figs. 7.18 and 7.19.

Connect the placed blocks together (Fig. 7.20).

Add the input DC source and a resistor to the schematic (Fig. 7.21). The resistor simulates the effect of winding wire resistance. The input DC voltage source is 6 V, since a 6-V voltage source provides 0.6 A in the steady state.

Add Probe and Scope blocks to the schematic in order to monitor the air gap flux density (Fig. 7.22).

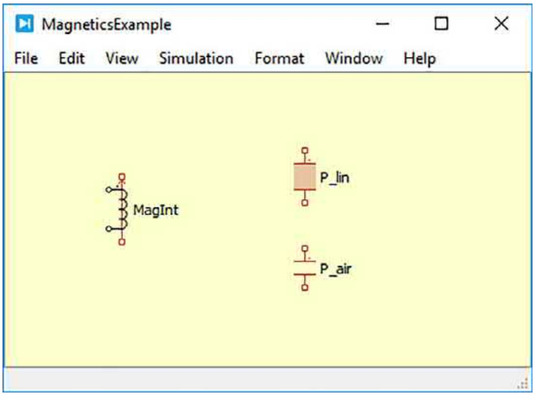
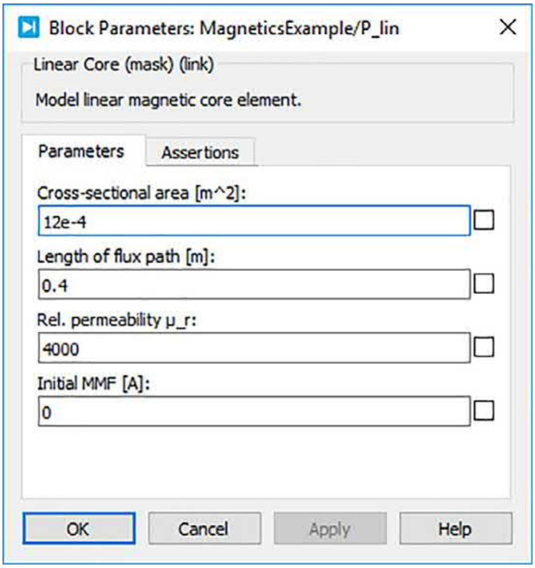

MagneticsExample
File Edit View Simulation Format Window Help
MagInt
P_lin
P_air

FIG. 7.17 Addition of Linear Core and Air Gap blocks to the simulation file.



Block Parameters: MagneticsExample/P_lin

Linear Core (mask) (link)

Model linear magnetic core element.

Parameters
Assertions

Cross-sectional area [m^2]: 12e-4 ☐

Length of flux path [m]: 0.4 ☐

Rel. permeability μ_r: 4000 ☐

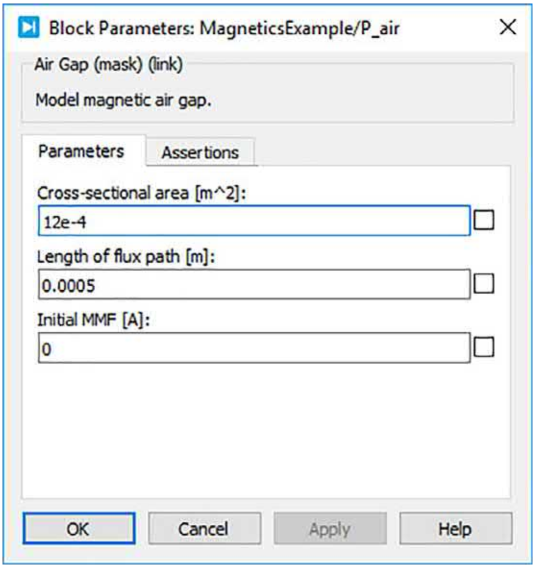
Initial MMF [A]: 0 ☐

OK
Cancel
Apply
Help

FIG. 7.18 Settings of Linear Core block.

Drag and drop the Air Gap block onto the Probe block (Fig. 7.23).

After the drag and drop, the Probe Editor window will appear. Select the Flux density (Fig. 7.24).



Block Parameters: MagneticsExample/P_air

Air Gap (mask) (link)

Model magnetic air gap.

Cross-sectional area [m^2]: 12e-4 ☐

Length of flux path [m]: 0.0005 ☐

Initial MMF [A]: 0 ☐

Parameters
Assertions
OK
Cancel
Apply
Help

FIG. 7.19 Setting of Air Gap block.

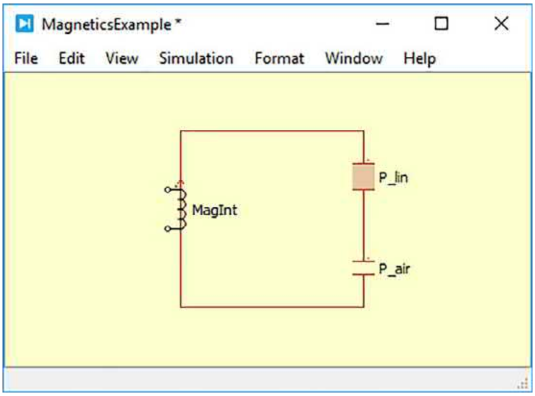

MagneticsExample *
File
Edit
View
Simulation
Format
Window
Help
MagInt
P_lin
P_air

FIG. 7.20 Components are connected together. When you move the mouse pointer behind the magnetic components terminals, the mouse pointer changes to crosshair, which lets you connect the terminal to other terminals.

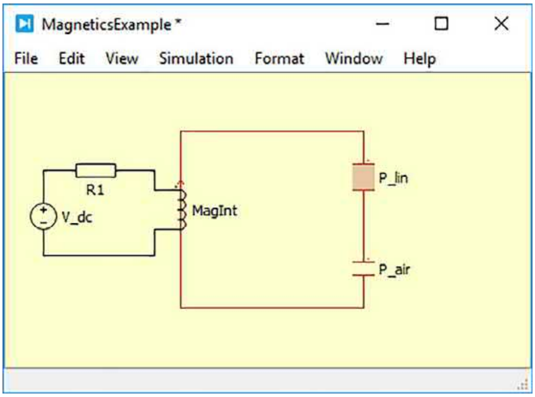

MagneticsExample *
File
Edit
View
Simulation
Format
Window
Help
R1
V_dc
MagInt
P_lin
P_air

FIG. 7.21 Addition of input DC source and a resistor. The resistor simulates the copper loss of windings.

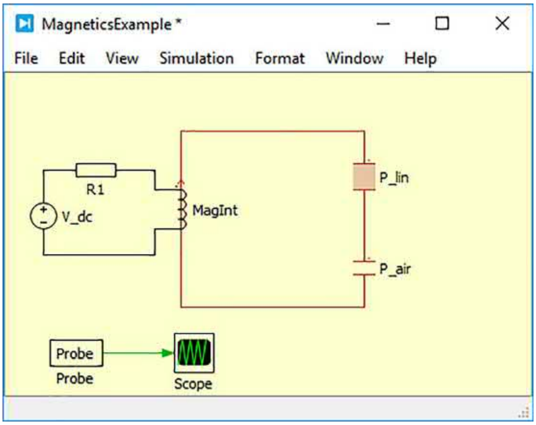

MagneticsExample *
File
Edit
View
Simulation
Format
Window
Help
R1
V_dc
MagInt
P_lin
P_air
Probe
Probe
Scope

FIG. 7.22 Signals of the circuit are monitored with the aid of Probe block.

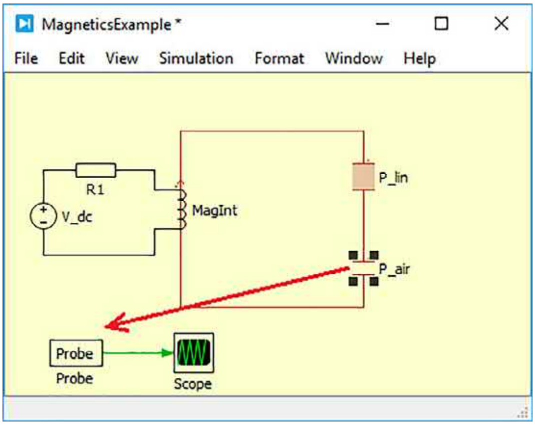
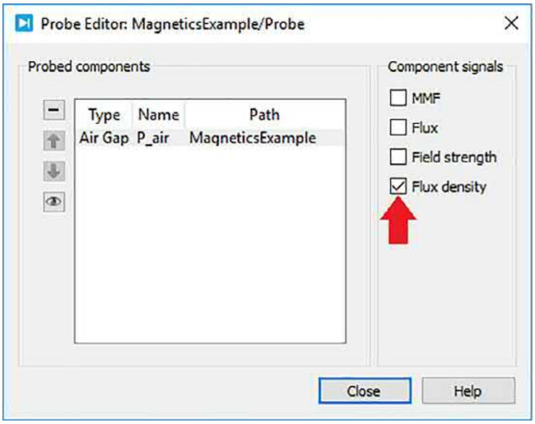

MagneticsExample *
File
Edit
View
Simulation
Format
Window
Help
R1
V_dc
MagInt
P_lin
P_air
Probe
Probe
Scope

FIG. 7.23 Drag the Air Gap block onto the Probe block.

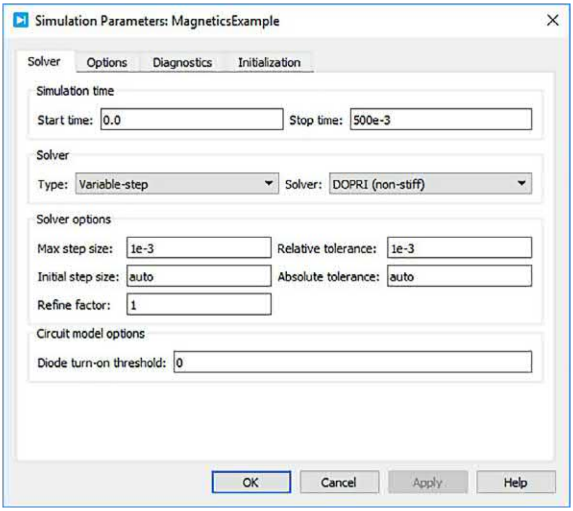

Probe Editor: MagneticsExample/Probe
Probed components
Component signals
Close
Help

Type	Name	Path
Air Gap	P_air	MagneticsExample

☐ MMF
☐ Flux
☐ Field strength
☑ Flux density

FIG. 7.24 The Probe Editor block.

Run the simulation with the settings shown in Fig. 7.25. The simulation result is shown in Fig. 7.26. The steady-state flux density of air gap is about 0.5 T.


Simulation Parameters: MagneticsExample
Solver
Options
Diagnostics
Initialization
Simulation time
Solver
Solver options
Circuit model options
OK
Cancel
Apply
Help

Start time: 0.0
Stop time: 500e-3
Type: Variable-step
Solver: DOPRI (non-stiff)
Max step size: 1e-3
Relative tolerance: 1e-3
Initial step size: auto
Absolute tolerance: auto
Refine factor: 1
Diode turn-on threshold: 0

FIG. 7.25 The simulation is done with the shown settings.

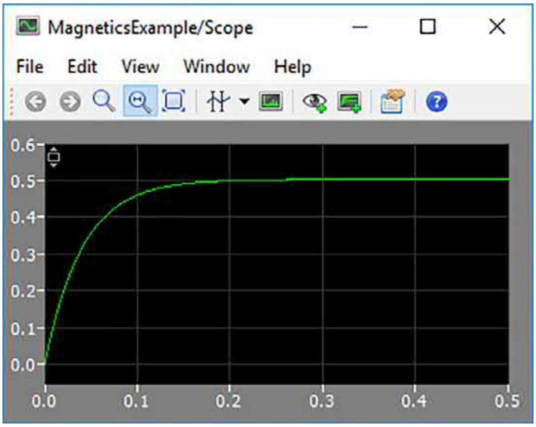


FIG. 7.26 Flux density in the air gap vs. time.

You can see the circuit current as well. In order to see the circuit current, add another Probe block to the schematic. Connect the placed Probe block to a Scope block (Fig. 7.27). Drag and drop the resistor R1 onto the Probe block. When Probe Editor appears, select the Resistor current.

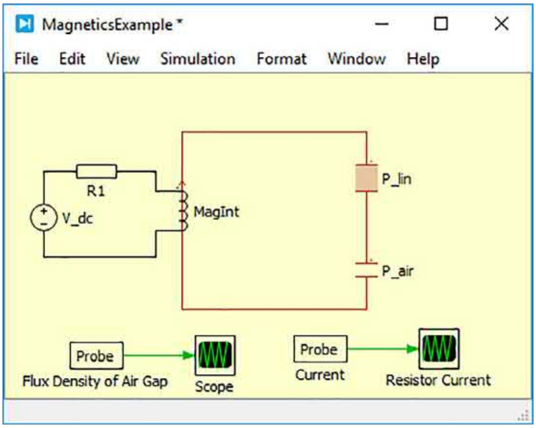

MagneticsExample *
File Edit View Simulation Format Window Help
R1
V_dc
MagInt
P_lin
P_air
Probe
Flux Density of Air Gap
Scope
Probe
Current
Resistor Current

FIG. 7.27 A Probe block is added to the simulation file in order to monitor the current in the winding.

The resistor current is shown in Fig. 7.28. We can use this Fig. in order to calculate the inductance of the configuration shown in Fig. 7.14. The time constant of current can be found by finding the time where the current reaches 0.63 of final value. The final value is 0.6 A. So, we search for the point, which the current reaches $0.63 \times 0.6 = 0.378$ A. At $t \approx 0.0396$ s, the current reaches 0.378 A (cursors are used to find the time point, which current reaches 0.378 A). So, according to $\tau = \frac{L}{R}$, the inductance is $L = 0.396$ H. Calculated value is quite close to the hand calculations.

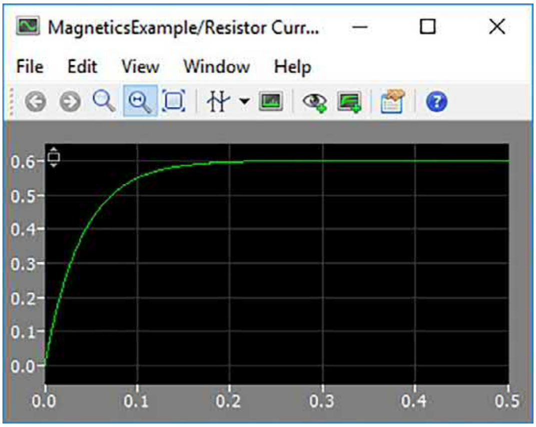


FIG. 7.28 The current inside the winding.

You can use the MMF Meter block to measure the magnetomotive forces. Two MMF Meter blocks are connected to Linear Core and Air Gap blocks to measure their magnetomotive forces (Fig. 7.29).

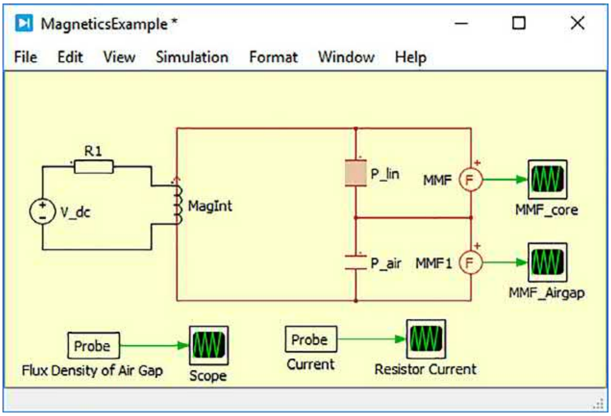

MagneticsExample *
File Edit View Simulation Format Window Help
R1
V_dc
MagInt
P_lin
MMF
MMF_core
P_air
MMF1
MMF_Airgap
Probe
Flux Density of Air Gap
Scope
Probe
Current
Resistor Current

FIG. 7.29 The magnetomotive forces can be measured with the aid of MMF Meter block.

Measured magnetomotive forces are shown in Figs. 7.30 and 7.31. The steady-state values of magnetomotive forces are the same as hand calculations.

The Flux Rate Meter measures the rate of change of magnetic flux ($\dot{\varphi}$) through the components. We can use this block to measure the voltage induced across the windings. According to Faraday's law of induction, $v = N\frac{d\varphi}{dt}$. The $\frac{d\varphi}{dt}$ is measured with the aid of Flux Rate Meter. So, if we multiply the output of Flux Rate Meter by the number of turns, we obtain the induced voltage across the windings. The schematic shown in Fig. 7.32 uses this technique to measure the induced voltage. The induced voltage is measured with the aid of a volt meter as well (Fig. 7.33). The output of two measurements is the same.

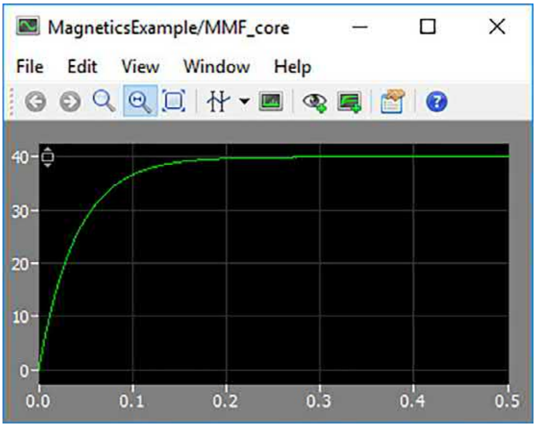


FIG. 7.30 Magnetomotive force across the Linear Core block.

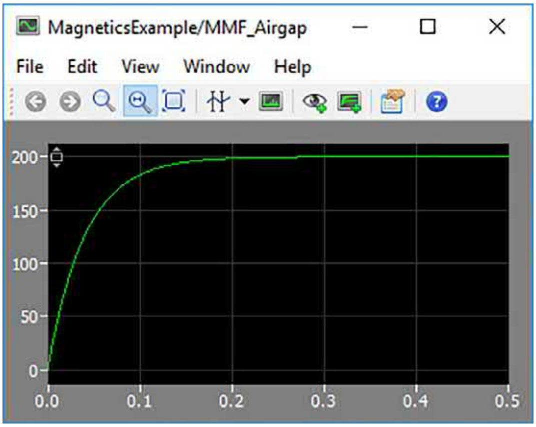


FIG. 7.31 Magnetomotive force across the Air Gap block.

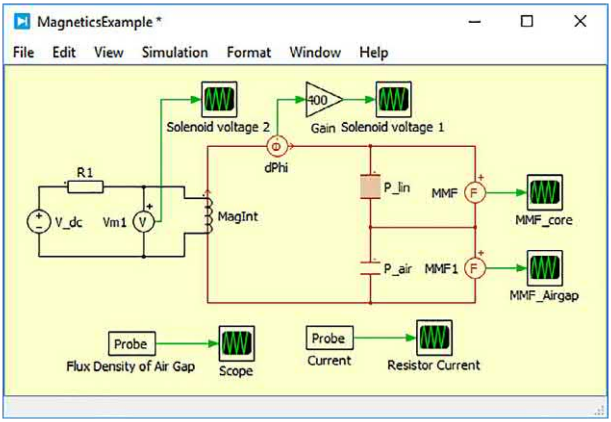

MagneticsExample *
File Edit View Simulation Format Window Help
Solenoid voltage 2
400
Gain
Solenoid voltage 1
dPhi
R1
V_dc
Vm1
MagInt
P_lin
MMF
MMF_core
P_air
MMF1
MMF_Airgap
Probe
Flux Density of Air Gap
Scope
Probe
Current
Resistor Current

FIG. 7.32 Measurement of voltage induced across the winding.

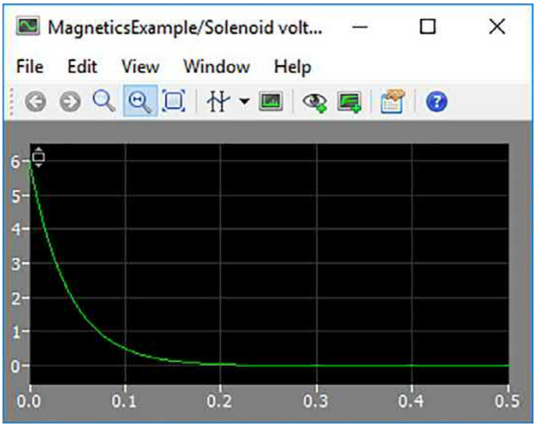
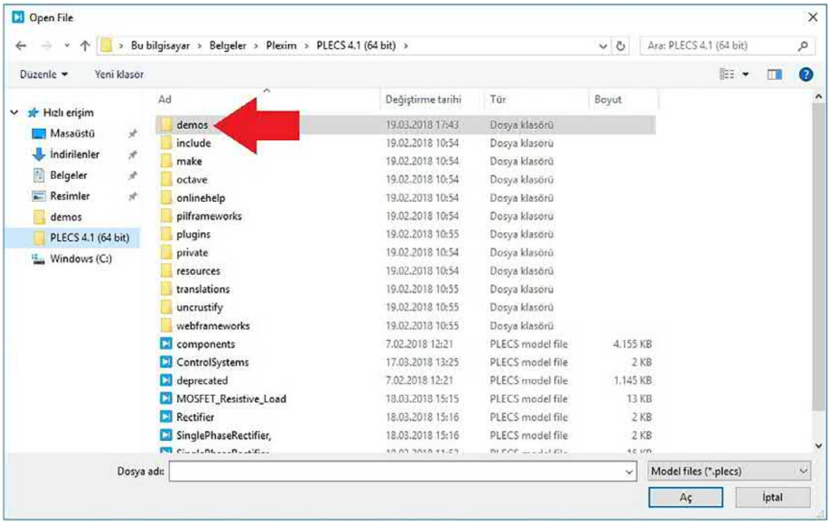


FIG. 7.33 Graph of produced voltage across the winding.

7.6 Where to go next?

PLECS® and world of power electronics simulation seems endless. If you followed the book up to this point, you learned the fundamentals and you can continue. One logical question is: "Where to go next?" Here we try to give you some idea.

PLECS® has a folder named demos (Fig. 7.34). It contains plenty of simulations. It can be a good source of inspiration.


Open File
Bu bilgisayar › Belgeler › Plexim › PLECS 4.1 (64 bit) ›
Ara: PLECS 4.1 (64 bit)
Düzenle
Yeni klasör
Hızlı erişim
Masaüstü
İndirilenler
Belgeler
Resimler
demos
PLECS 4.1 (64 bit)
Windows (C:)
Dosya adı:
Model files (*.plecs)
Aç
İptal

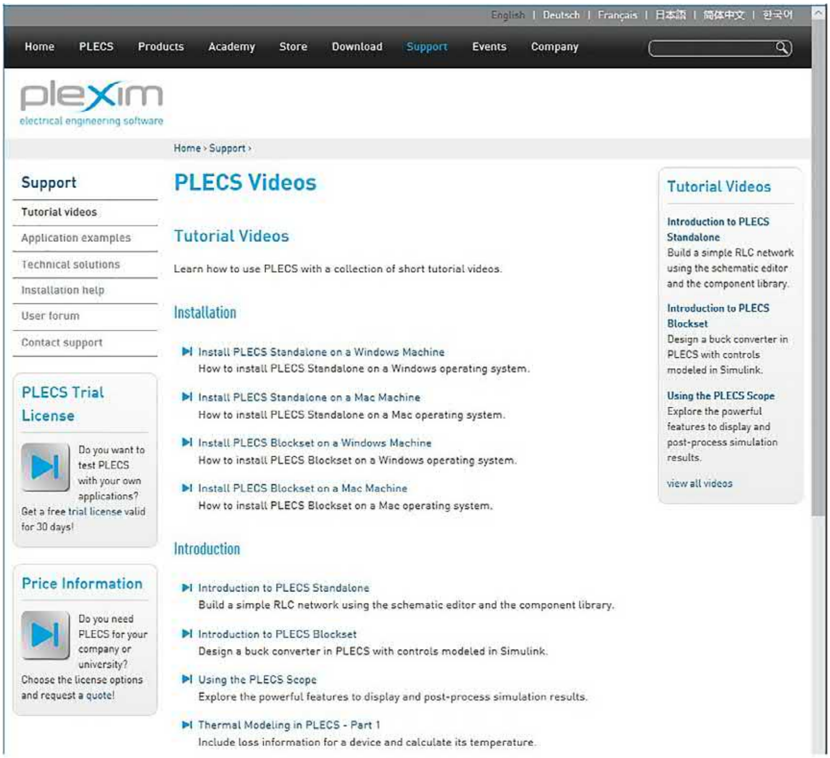
Ad	Değiştirme tarihi	Tür	Boyut
demos	19.03.2018 17:43	Dosya klasörü	
include	19.02.2018 10:54	Dosya klasörü	
make	19.02.2018 10:54	Dosya klasörü	
octave	19.02.2018 10:54	Dosya klasörü	
onlinehelp	19.02.2018 10:54	Dosya klasörü	
pilframeworks	19.02.2018 10:54	Dosya klasörü	
plugins	19.02.2018 10:55	Dosya klasörü	
private	19.02.2018 10:54	Dosya klasörü	
resources	19.02.2018 10:54	Dosya klasörü	
translations	19.02.2018 10:55	Dosya klasörü	
uncrustify	19.02.2018 10:55	Dosya klasörü	
webframeworks	19.02.2018 10:55	Dosya klasörü	
components	7.02.2018 12:21	PLECS model file	4.155 KB
ControlSystems	17.03.2018 13:25	PLECS model file	2 KB
deprecated	7.02.2018 12:21	PLECS model file	1.145 KB
MOSFET_Resistive_Load	18.03.2018 15:15	PLECS model file	13 KB
Rectifier	18.03.2018 15:16	PLECS model file	2 KB
SinglePhaseRectifier,	18.03.2018 15:16	PLECS model file	2 KB

FIG. 7.34 PLECS comes with plenty of demo simulations placed at the demo folder.

Visit PLECS® website regularly. It contains tutorial videos, academic programs, news, etc. (Figs. 7.35 and 7.36).

Do not forget to refer to PLECS® documentation (Fig. 7.37).

Visit "User forum" to see the question and answers about the PLECS®. You can even ask your questions there (Fig. 7.38)


English | Deutsch | Français | 日本語 | 簡体中文 | 한국어
Home
PLECS
Products
Academy
Store
Download
Support
Events
Company
plexim
electrical engineering software
Home › Support ›
Support
Tutorial videos
Application examples
Technical solutions
Installation help
User forum
Contact support

PLECS Videos

Tutorial Videos

Learn how to use PLECS with a collection of short tutorial videos.

Installation

- Install PLECS Standalone on a Windows Machine
  How to install PLECS Standalone on a Windows operating system.
- Install PLECS Standalone on a Mac Machine
  How to install PLECS Standalone on a Mac operating system.
- Install PLECS Blockset on a Windows Machine
  How to install PLECS Blockset on a Windows operating system.
- Install PLECS Blockset on a Mac Machine
  How to install PLECS Blockset on a Mac operating system.

Introduction

- Introduction to PLECS Standalone
  Build a simple RLC network using the schematic editor and the component library.
- Introduction to PLECS Blockset
  Design a buck converter in PLECS with controls modeled in Simulink.
- Using the PLECS Scope
  Explore the powerful features to display and post-process simulation results.
- Thermal Modeling in PLECS - Part 1
  Include loss information for a device and calculate its temperature.

PLECS Trial License

Do you want to test PLECS with your own applications? Get a free trial license valid for 30 days!

Price Information

Do you need PLECS for your company or university? Choose the license options and request a quote!

Tutorial Videos

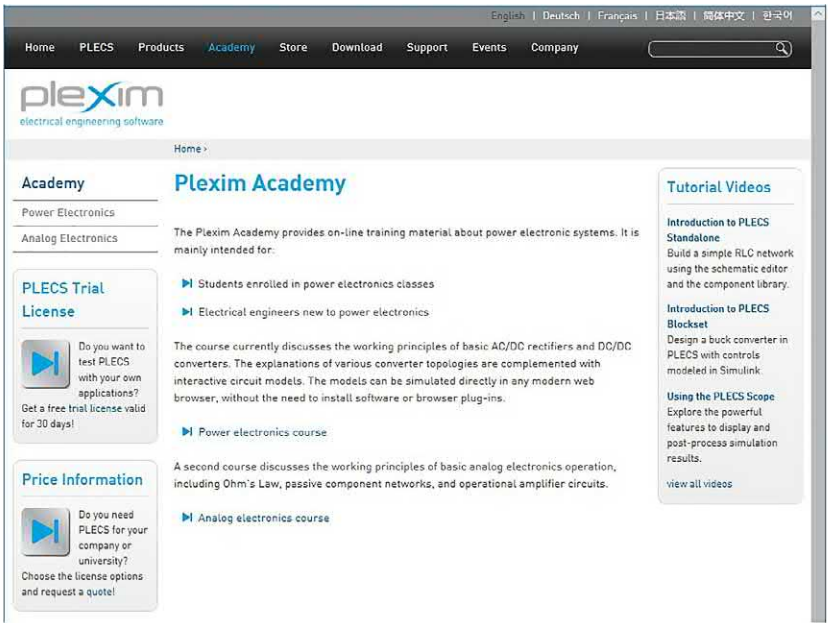
Introduction to PLECS Standalone
Build a simple RLC network using the schematic editor and the component library.

Introduction to PLECS Blockset
Design a buck converter in PLECS with controls modeled in Simulink.

Using the PLECS Scope
Explore the powerful features to display and post-process simulation results.

view all videos

FIG. 7.35 https://www.plexim.com/support/videos.


English | Deutsch | Français | 日本語 | 簡体中文 | 한국어
Home
PLECS
Products
Academy
Store
Download
Support
Events
Company
plexim
electrical engineering software
Home ›
Academy
Power Electronics
Analog Electronics

Plexim Academy

The Plexim Academy provides on-line training material about power electronic systems. It is mainly intended for:

- Students enrolled in power electronics classes
- Electrical engineers new to power electronics

The course currently discusses the working principles of basic AC/DC rectifiers and DC/DC converters. The explanations of various converter topologies are complemented with interactive circuit models. The models can be simulated directly in any modern web browser, without the need to install software or browser plug-ins.

- Power electronics course

A second course discusses the working principles of basic analog electronics operation, including Ohm's Law, passive component networks, and operational amplifier circuits.

- Analog electronics course

PLECS Trial License

Do you want to test PLECS with your own applications? Get a free trial license valid for 30 days!

Price Information

Do you need PLECS for your company or university? Choose the license options and request a quote!

Tutorial Videos

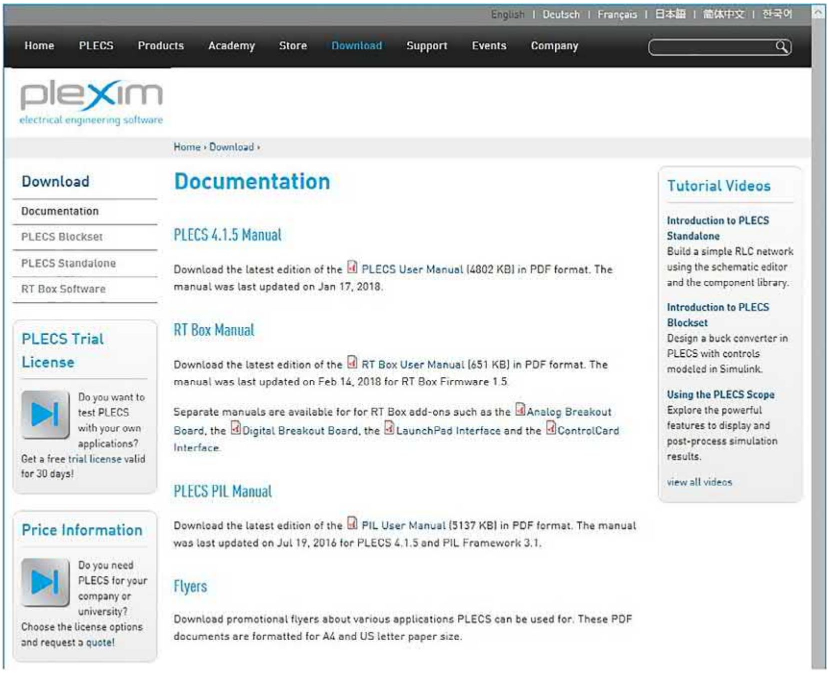
Introduction to PLECS Standalone
Build a simple RLC network using the schematic editor and the component library.

Introduction to PLECS Blockset
Design a buck converter in PLECS with controls modeled in Simulink.

Using the PLECS Scope
Explore the powerful features to display and post-process simulation results.

view all videos

FIG. 7.36 https://www.plexim.com/academy.


English | Deutsch | Français | 日本語 | 简体中文 | 한국어
Home
PLECS
Products
Academy
Store
Download
Support
Events
Company
plexim
electrical engineering software
Home › Download ›
Download
Documentation
PLECS Blockset
PLECS Standalone
RT Box Software

Documentation

PLECS 4.1.5 Manual

Download the latest edition of the PLECS User Manual (4802 KB) in PDF format. The manual was last updated on Jan 17, 2018.

RT Box Manual

Download the latest edition of the RT Box User Manual (651 KB) in PDF format. The manual was last updated on Feb 14, 2018 for RT Box Firmware 1.5.

Separate manuals are available for for RT Box add-ons such as the Analog Breakout Board, the Digital Breakout Board, the LaunchPad Interface and the ControlCard Interface.

PLECS PIL Manual

Download the latest edition of the PIL User Manual (5137 KB) in PDF format. The manual was last updated on Jul 19, 2016 for PLECS 4.1.5 and PIL Framework 3.1.

Flyers

Download promotional flyers about various applications PLECS can be used for. These PDF documents are formatted for A4 and US letter paper size.

PLECS Trial License

Do you want to test PLECS with your own applications? Get a free trial license valid for 30 days!

Price Information

Do you need PLECS for your company or university? Choose the license options and request a quote!

Tutorial Videos

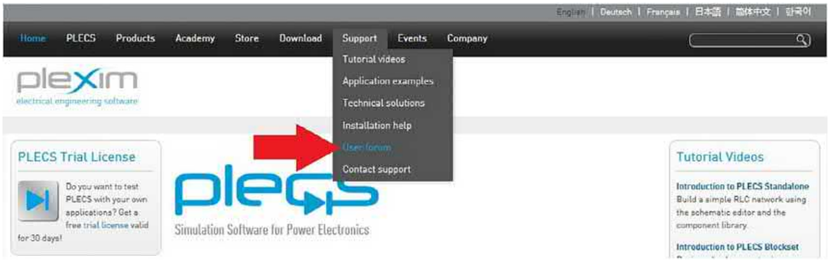
Introduction to PLECS Standalone
Build a simple RLC network using the schematic editor and the component library.

Introduction to PLECS Blockset
Design a buck converter in PLECS with controls modeled in Simulink.

Using the PLECS Scope
Explore the powerful features to display and post-process simulation results.

view all videos

FIG. 7.37 https://www.plexim.com/download/documentation.


English | Deutsch | Français | 日本語 | 简体中文 | 한국어
Home
PLECS
Products
Academy
Store
Download
Support
Events
Company
Tutorial videos
Application examples
Technical solutions
Installation help
User forum
Contact support
plexim
electrical engineering software
PLECS
Simulation Software for Power Electronics

PLECS Trial License

Do you want to test PLECS with your own applications? Get a free trial license valid for 30 days!

Tutorial Videos

Introduction to PLECS Standalone
Build a simple RLC network using the schematic editor and the component library.

Introduction to PLECS Blockset

FIG. 7.38 PLECS$^{®}$ website has a user forum section.

Reference

[1] S.J. Chapman, Electric Machinery Fundamentals, McGraw Hill, 2011.

Further reading

[2] K. Watanabe, F. Campelo, Y. Iijima, K. Kawano, T. Matsuo, T. Mifune, H. Igarash, Optimization of inductors using evolutionary algorithms and its experimental validation, IEEE Trans. Magnet. 46 (8) (2010)August.

Chapter 8

Fundamental concepts of power electronic circuits

Chapter outline

8.1 Introduction 421
8.2 Instantaneous power 422
8.3 Average power 424
8.4 Effective value of a signal 425
  8.4.1 Effective value of sum of two periodic signals 429
  8.4.2 Measurement of RMS of signals 431
8.5 Apparent power and power factor 432
8.6 Power computations for linear circuits 433
8.7 Fourier series 434
  8.7.1 Fourier series of important waveshapes 435
  8.7.2 Calculation of average power using the Fourier series 437
8.8 Total harmonic distortion (THD) 438
8.9 Volt-second balance 439
8.10 Ampere-second balance 440
8.11 MOSFET with resistive load 441
8.12 Uncontrolled half-wave rectifier 443
8.13 Controlled half-wave rectifier 444
8.14 DC-DC converters 445
  8.14.1 Buck converter 446
8.15 Calculation of output voltage of a buck converter operated in DCM 452
8.16 Other types of DC-DC converters operating in DCM 454
  8.16.1 Boost converter 454
  8.16.2 Buck-boost converter 455
  8.16.3 Cuk converter 455
  8.16.4 Flyback converter 456
8.17 Dynamics of DC-DC converters 457
  8.17.1 Overview of state space averaging (SSA) 457
  8.17.2 Dynamical model of buck converter 459
8.18 PID controller design for converter 474
8.19 Input/output impedance of converter 478
8.20 Effect of feedback control on output impedance 486
8.21 Dynamic of buck-boost converter 488
8.22 Dynamics of boost converter 495
8.23 Dynamics of zeta converter 502
8.24 Inverters 515
  8.24.1 Series H bridge inverters 518
  8.24.2 Diode-clamped multilevel inverters 519
8.25 Heatsink 521
Further reading 526

Simulation of Power Electronics Converters Using PLECS®. https://doi.org/10.1016/B978-0-12-817364-0.00008-4
© 2020 Elsevier Inc. All rights reserved.

8.1 Introduction

In the previous chapters, we showed how PLECS® can be used to analyze different power electronics converters. This is the last chapter of book and it will study some of the important theories of power electronics circuits. We focus on theories, which help the reader understand the example circuits of previous chapters better. Plenty of good power electronics references are available in the market. [1] is a good textbook with plenty of numeric examples. It is quite useful for a first course on power electronics, since the concepts are described clearly. [2-4] are more advanced books and suggested to advanced readers. [5] and [6] are good references for readers interested in DC-DC converters. [7] is a good reference for readers interested in inverters.

It is a good idea to do some hand analysis for the circuit drawn in the simulator environment and compare the simulation result with hand analysis. If the results are different, try to find the source of discrepancy. For instance, when someone analyzes a half-wave rectifier, generally the voltage drop of diodes is neglected. So, the simulation result (which considers the voltage drop of diodes) is not the same as hand analysis. Such practices help you to understand the power electronics concepts better.

8.2 Instantaneous power

The instantaneous power of a device ($p(t)$) is defined as:

$$p(t) = v(t) \times i(t), \tag{8.1}$$

where $v(t)$ is the voltage across the device and $i(t)$ is the current through the device. The instantaneous power is generally a time-varying quantity. If the passive sign convention illustrated in Fig. 8.1A is observed, the device is absorbing power if $p(t)$ is positive at a specified value of time t. The device is supplying power if $p(t)$ is negative.

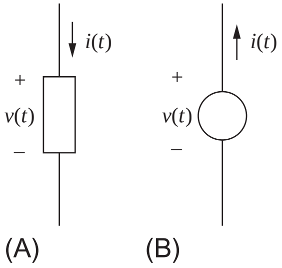


FIG. 8.1 (A) Passive sign convention: $p(t) > 0$ indicates power is being absorbed; (B) $p(t) > 0$ indicates power is being supplied by the source.

For instance, consider the simple circuit shown in Fig. 8.2. In this circuit, $v_{in}(t) = 311\sin(377t)$ and $R = 50\ \Omega$.

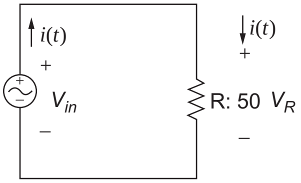


FIG. 8.2 A simple resistive circuit.

According to Ohm's law, $i(t) = \frac{v_{in}(t)}{R} = 6.22\sin(377t)$ and instantaneous power for resistor R is:

$$p_R(t) = 311\sin(377t) \times 6.22\sin(377t) = 2345\sin^2(377t). \tag{8.2}$$

Obtained result is positive for all the times, i.e., $\forall t$, $\sin^2(377t) > 0$. This is expected since resistor dissipates power.

The instantaneous power of AC source can be calculated with the aid of Fig. 8.3.

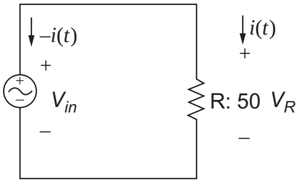


FIG. 8.3 Calculation of instantaneous power of input AC source.

The instantaneous power of AC source is:

$$p_{V_{in}}(t) = 311\sin(377t) \times -6.22\sin(377t) = -2345\sin^2(377t) \tag{8.3}$$

Obtained result is negative for all the time. We expect this result since the AC source supplies the power into the load. For instance at $t = 12$ ms, $p_{V_{in}}(t) = -2.263$ kW, and $p_R(t) = +2.263$ kW. This means that at $t = 12$ ms, AC source supplies 2.263 kW and resistor absorbs 2.263 kW. Fig. 8.4 shows the instantaneous power waveforms on the same graph.

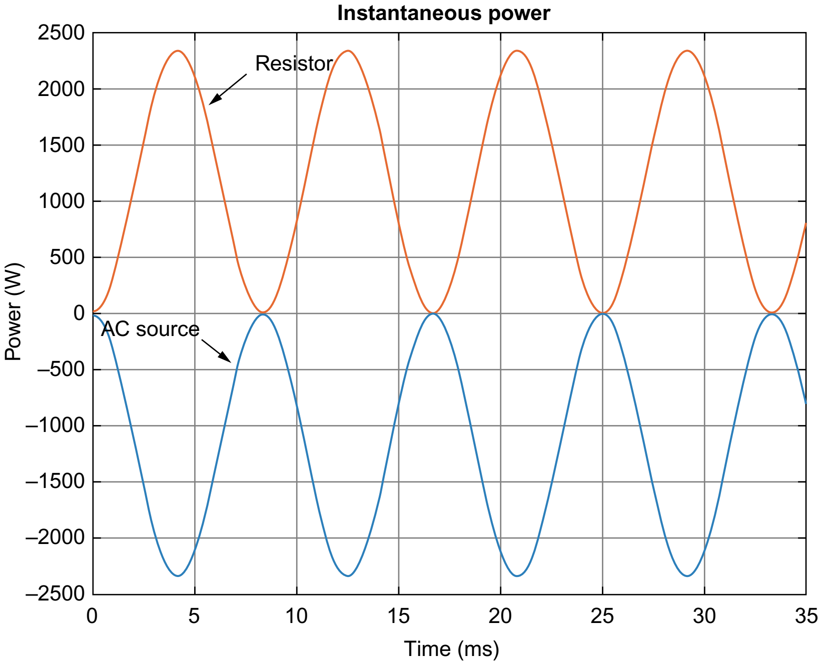


FIG. 8.4 Graph of instantaneous power for input AC source and load resistor. The instantaneous power of AC source is negative since it supplies power to the resistor. The resistor instantaneous power is positive since it consumes power.

8.3 Average power

Function $f(t)$ is periodic if,

$$\exists T > 0, \forall t f(t+T) = f(t) \tag{8.4}$$

T is called the period. For instance, $f(t) = \sin(t)$ is periodic since $f(t+2\pi) = f(t)$.

If device voltage ($v(t)$) and current ($i(t)$) are periodic, i.e., $v(t) = v(t+T)$ and $i(t) = i(t+T)$, then the instantaneous power will be a periodic since

$$\begin{gathered} p(t) = v(t) \times i(t) \\ p(t+T) = v(t+T) \times i(t+T) = v(t) \times i(t) = p(t) \end{gathered} \tag{8.5}$$

The average power for such a periodic waveform is defined as:

$$P = \frac{1}{T}\int_{t_0}^{t_0+T} p(t)dt = \frac{1}{T}\int_{t_0}^{t_0+T} v(t) \times i(t)dt \tag{8.6}$$

Assume that $v(t)$ is a constant function, i.e., $v(t) = V_{dc}$. In this case, the average power can be calculated by the

$$P = \frac{1}{T}\int_{t_0}^{t_0+T} v(t) \times i(t)dt = \frac{1}{T}\int_{t_0}^{t_0+T} V_{dc} \times i(t)dt = V_{dc}\left[\frac{1}{T}\int_{t_0}^{t_0+T} i(t)dt\right] = V_{dc}I_{avg} \tag{8.7}$$

The average power for constant $i(t)$, i.e., $i(t)=I_{dc}$ can be found in the same way.

$$P=\frac{1}{T}\int_{t_0}^{t_0+T} v(t)\times i(t)dt=\frac{1}{T}\int_{t_0}^{t_0+T} v(t)\times I_{dc}dt=I_{dc}\left[\frac{1}{T}\int_{t_0}^{t_0+T} v(t)dt\right]=I_{dc}v_{avg} \tag{8.8}$$

8.4 Effective value of a signal

Consider the simple circuit shown in Fig. 8.5. The input source is a periodic voltage source, i.e., $v(t+T)=v(t)$. The load is purely resistive.

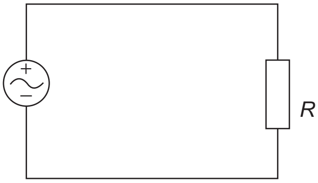


FIG. 8.5 A resistor is connected to a periodic voltage source.

The power consumed by the resistor is:

$$P=\frac{1}{T}\int_0^T p(t)dt=\frac{1}{T}\int_0^T v(t)\times i(t)dt=\frac{1}{T}\int_0^T \frac{v(t)^2}{R}dt=\frac{1}{R}\left[\frac{1}{T}\int_0^T v(t)^2dt\right] \tag{8.9}$$

Now consider the circuit shown in Fig. 8.6. The input source is a constant DC voltage source, i.e., $v(t)=V_{dc}$.

FIG. 8.6 The same resistor (as the one in Fig. 8.5) is connected to a DC source.

In this case, the power consumed by the resistor is $\frac{V_{dc}^2}{R}$. Power consumption of both circuits is the same when $V_{dc}=\sqrt{\frac{1}{T}\int_0^T v(t)^2dt}$. Since,

$$\frac{1}{R}\left[\frac{1}{T}\int_0^T v(t)^2dt\right]=\frac{V_{dc}^2}{R}\Longrightarrow V_{dc}=\sqrt{\frac{1}{T}\int_0^T v(t)^2dt} \tag{8.10}$$

The $\sqrt{\frac{1}{T}\int_0^T v(t)^2dt}$ is called Root Mean Square (RMS) or effective value of signal $v(t)$. So, RMS value of periodic signal $v(t)$ is a DC value, which produces the same amount of heat in the resistive load as the periodic signal $v(t)$.

The RMS can be defined for the current waveforms as well.

$$I_{rms} = \sqrt{\frac{1}{T}\int_0^T i(t)^2dt} \tag{8.11}$$

Example 8.1 Determine the rms value of the periodic pulse waveform shown in Fig. 8.7.

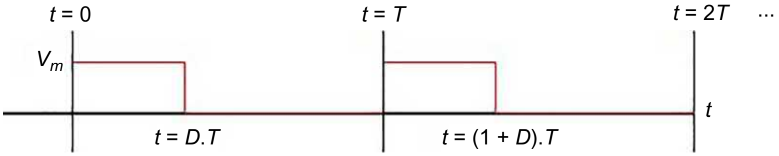


FIG. 8.7 Waveform of Example 8.1.

Solution

$$v(t) = \begin{cases} V_m & 0 < t < DT \\ 0 & DT < t < T \end{cases}$$

$$V_{rms} = \sqrt{\frac{1}{T}\int_0^T v(t)^2dt} = \sqrt{\frac{1}{T}\left(\int_0^{DT} V_m{}^2dt + \int_{DT}^T 0dt\right)} = \sqrt{\frac{1}{T}\left(V_m{}^2DT\right)} = V_m\sqrt{D}$$

Example 8.2 Determine the RMS values of the following waveforms $\left(\omega = \frac{2\pi}{T}\right)$.

(a) $v(t) = V_m\sin(\omega t)$.
(b) $v(t) = |V_m\sin(\omega t)|$.
(c) $v(t) = \begin{cases} V_m\sin(\omega t) & 0 < t < \dfrac{T}{2} \\ 0 & \dfrac{T}{2} < t < T \end{cases}$.

Solution

(a)

$$V_{rms} = \sqrt{\frac{1}{T}\int_0^T (V_m \sin(\omega t))^2 dt} = \sqrt{\frac{1}{T} \times V_m{}^2 \int_0^T \sin^2(\omega t) dt} = \sqrt{\frac{V_m{}^2}{T}\int_0^T \frac{1-\cos(2\omega t)}{2} dt}$$

$$= \sqrt{\frac{V_m{}^2}{T}\int_0^T \frac{1}{2} dt - \int_0^T \frac{\cos(2\omega t)}{2} dt}$$

$$= \sqrt{\frac{V_m{}^2}{T} \times \left(\frac{T}{2} - \frac{\sin(2\omega t)}{4\omega}\right)\Big|_0^T}$$

$$= \sqrt{\frac{V_m{}^2}{T} \times \frac{T}{2} - 0}$$

$$= \sqrt{\frac{V_m{}^2}{2}}$$

$$= \frac{V_m}{\sqrt{2}}$$

(b) RMS value of $v(t) = |V_m \sin(\omega t)|$ is the same as $v(t) = V_m \sin(\omega t)$. Since $(|V_m \sin(\omega t)|)^2 = (V_m \sin(\omega t))^2$. So, RMS value of $v(t) = |V_m \sin(\omega t)|$ is $\frac{V_m}{\sqrt{2}}$. Graph of $v(t) = |V_m \sin(\omega t)|$ is shown in Fig. 8.8. Such a waveform is called Full-Wave Rectified in power electronics.

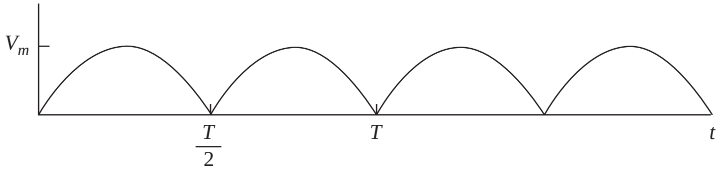


FIG. 8.8 Full-wave rectified sinusoidal waveform.

(c) Graph of $v(t) = \begin{cases} V_m \sin(\omega t) & 0 < t < \frac{T}{2} \\ 0 & \frac{T}{2} < t < T \end{cases}$ is shown in Fig. 8.9. Such a waveform is called Half-Wave Rectified in power electronics.

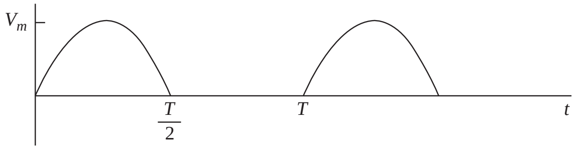

V_m
$\frac{T}{2}$
T
t

FIG. 8.9 Half-wave rectified sinusoidal waveform.

$$V_{rms} = \sqrt{\frac{1}{T}\left(\int_0^{\frac{T}{2}} (V_m \sin(\omega t))^2 dt + \int_{\frac{T}{2}}^{T} 0dt\right)} = \sqrt{\frac{1}{T} \times {V_m}^2 \int_0^{\frac{T}{2}} \sin^2(\omega t) dt}$$

$$= \sqrt{\frac{{V_m}^2}{T} \int_0^{\frac{T}{2}} \frac{1-\cos(2\omega t)}{2} dt} = \sqrt{\frac{{V_m}^2}{T} \int_0^{\frac{T}{2}} \frac{1}{2} dt - \int_0^{\frac{T}{2}} \frac{\cos(2\omega t)}{2} dt}$$

$$= \sqrt{\frac{{V_m}^2}{T} \times \left(\frac{t}{2} - \frac{\sin(2\omega t)}{4\omega}\right)\Big|_0^{\frac{T}{2}}}$$

$$= \sqrt{\frac{{V_m}^2}{T} \times \frac{T}{4} - 0}$$

$$= \sqrt{\frac{{V_m}^2}{4}}$$

$$= \frac{V_m}{2}$$

RMS of triangular waveshapes can be calculated using the formulas shown in Fig. 8.10.

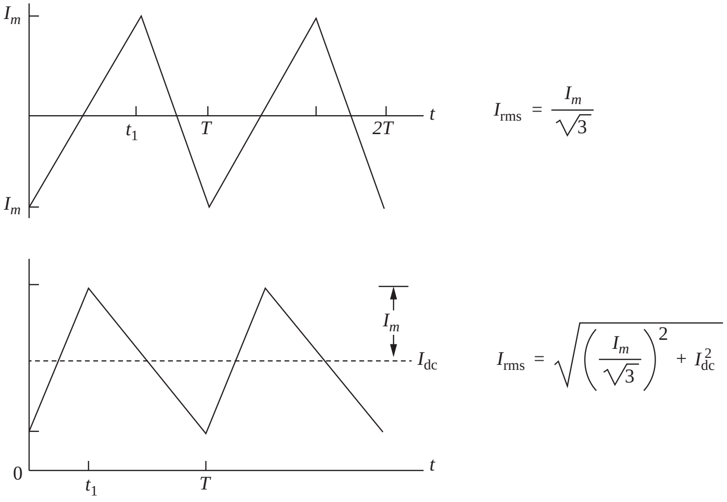

I_m
I_m
t_1
T
$2T$
t
$I_{rms} = \frac{I_m}{\sqrt{3}}$
I_m
I_{dc}
0
t_1
T
t
$I_{rms} = \sqrt{\left(\frac{I_m}{\sqrt{3}}\right)^2 + I_{dc}^2}$

FIG. 8.10 RMS value of triangular waveforms.

8.4.1 Effective value of sum of two periodic signals

Consider two periodic waveforms, i.e., $v_1(t+T)=v_1(t)$, $v_2(t+T)=v_2(t)$. The RMS value of sum of two waveforms ($v(t)=v_1(t)+v_2(t)$) is:

$$\begin{aligned}{V_{rms}}^2&=\frac{1}{T}\int_0^T(v_1+v_2)^2dt=\frac{1}{T}\int_0^T\left({v_1}^2+2v_1v_2+{v_2}^2\right)dt\\&=\frac{1}{T}\int_0^T{v_1}^2dt+\frac{1}{T}\int_0^T 2v_1v_2dt+\frac{1}{T}\int_0^T{v_2}^2dt\end{aligned}\tag{8.12}$$

Sometime, the $\frac{1}{T}\int_0^T v_1(t)v_2(t)dt$ term is zero. The $\frac{1}{T}\int_0^T v_1(t)v_2(t)dt$ is the inner product of $v_1(t)$ and $v_2(t)$. When $\frac{1}{T}\int_0^T v_1(t)v_2(t)dt=0$, the signals $v_1(t)$ and $v_2(t)$ are called orthogonal. Table 8.1 shows some of the important orthogonal functions.

TABLE 8.1 Some of the important orthogonal functions ($\omega=\frac{2\pi}{T}$, $n\neq m$, and k is a constant).

No	$v_1(t)$	$v_2(t)$
1	$\sin(n\times\omega\times t+\varphi_1)$	$\sin(m\times\omega\times t+\varphi_2)$
2	$\sin(n\times\omega\times t+\varphi_1)$	$\cos(m\times\omega\times t+\varphi_2)$
3	$\cos(n\times\omega\times t+\varphi_1)$	$\cos(m\times\omega\times t+\varphi_2)$
4	$\sin(n\times\omega\times t+\varphi_1)$	k
5	$\cos(m\times\omega\times t+\varphi_1)$	k

For instance according to the second row of the table, $\sin(n\times\omega\times t+\varphi_1)$ and $\cos(m\times\omega\times t+\varphi_2)$ (when $n\neq m$) are orthogonal since $\frac{1}{T}\int_0^T\sin(n\omega t+\varphi_1)\times\cos(m\omega t+\varphi_2)dt=0$.

For orthogonal functions,

$${V_{rms}}^2=\frac{1}{T}\int_0^T(v_1+v_2)^2dt=\frac{1}{T}\int_0^T\left({v_1}^2+2v_1v_2+{v_2}^2\right)dt$$

$${V_{rms}}^2=\frac{1}{T}\int_0^T{v_1}^2dt+\frac{1}{T}\int_0^T 2v_1v_2dt+\frac{1}{T}\int_0^T{v_2}^2dt$$

$${V_{rms}}^2=\frac{1}{T}\int_0^T{v_1}^2dt+\frac{1}{T}\int_0^T{v_2}^2dt$$

$$V_{rms}=\sqrt{{V_{1,rms}}^2+{V_{2,rms}}^2}\tag{8.13}$$

RMS value of sum of more than two orthogonal functions (every two terms are assumed to be orthogonal) can be calculated in the same way:

$$\left(v(t)=\sum_{n=1}^{N}v_n(t)\forall k,l\,1\le k\le N,1\le l\le N,k\ne l,\ \frac{1}{T}\int_0^T v_k(t)v_l(t)dt=0\right)\Longrightarrow V_{rms}$$
$$=\sqrt{V_{1,rms}{}^2+V_{2,rms}{}^2+V_{3,rms}{}^2+\ldots}=\sqrt{\sum_{n=1}^{N}V_{n,rms}{}^2} \tag{8.14}$$

Example 8.3 Determine the RMS value of $v(t)=4+8\sin(\omega_1 t+10°)+5\sin(\omega_2 t+50°)$ under the following conditions.

(a) $\omega_2=2\omega_1$
(b) $\omega_2=\omega_1$

Solution

(a) When $\omega_2=2\omega_1$, the $v(t)=4+8\sin(\omega_1 t+10°)+5\sin(2\omega_1 t+50°)$. According to Table 8.1, all the functions are orthogonal to each other, so

$$V_{rms}=\sqrt{V_{1,rms}{}^2+V_{2,rms}{}^2+V_{3,rms}{}^2}=\sqrt{4^2+\left(\frac{8}{\sqrt{2}}\right)^2+\left(\frac{5}{\sqrt{2}}\right)^2}=7.78\,\mathrm{V}$$

(b) When $\omega_2=\omega_1$, the $v(t)=4+8\sin(\omega_1 t+10°)+5\sin(\omega_1 t+50°)$. $8\sin(\omega_1 t+10°)$ and $5\sin(\omega_1 t+50°)$ are not orthogonal to each other. So, we can't use the previous formulas. Note that $a\times\sin(\omega t)+b\times\cos(\omega t)=\sqrt{a^2+b^2}\sin\left(\omega t+\tan^{-1}\left(\frac{b}{a}\right)\right)$. So,

$$v(t)=4+8\sin(\omega_1 t+10°)+5\sin(\omega_1 t+50°)=4+12.3\sin(\omega_1 t+25.2°)$$

The two terms of last equation are orthogonal to each other (see Table 8.1). So, the RMS is

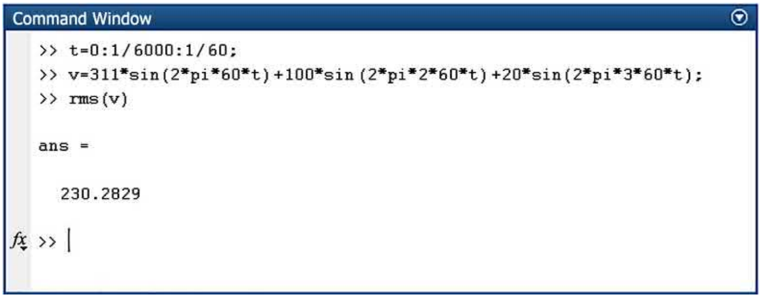
$$V_{rms}=\sqrt{4^2+\left(\frac{12.3}{\sqrt{2}}\right)^2}=9.57\,V$$

Example 8.4 In this example, we show how RMS values can be calculated with the aid of MATLAB®. Assume

$v(t)=311\sin(2\pi\times 60t)+100\sin(2\pi\times 2\times 60t)+20\sin(2\pi\times 3\times 60t)$ is given. The RMS can be calculated easily:

$$V_{rms}=\sqrt{\left(\frac{311}{\sqrt{2}}\right)^2+\left(\frac{100}{\sqrt{2}}\right)^2+\left(\frac{20}{\sqrt{2}}\right)^2}=231.43\,V$$

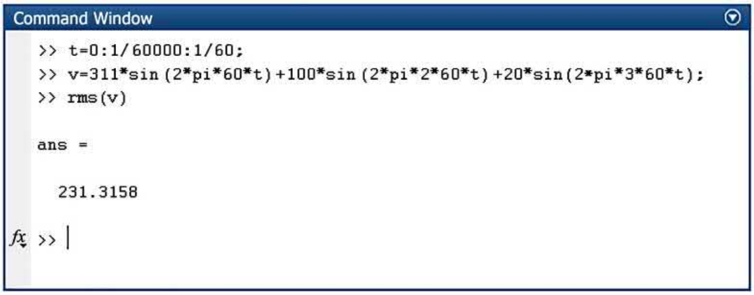
The commands shown in Fig. 8.11 calculate the RMS value of given signal. The first two lines sample a period of given signal. The sampling time is $\frac{1}{6000} = 166.7\mu s$. The `rms` command is used to calculate the RMS value of sampled signal.



```
>> t=0:1/6000:1/60;
>> v=311*sin(2*pi*60*t)+100*sin(2*pi*2*60*t)+20*sin(2*pi*3*60*t);
>> rms(v)

ans =

  230.2829

fx >> |
```

FIG. 8.11 Calculation of RMS value of $v(t) = 311\sin(2\pi \times 60t) + 100\sin(2\pi \times 2 \times 60t) + 20\sin(2\pi \times 3 \times 60t)$ with $\frac{1}{6000}$ steps.

The result is 230.283, which is a little bit lower than the expected value of 231.43. If you decrease the sampling time from 166.7 μs to 16.67 μs, you get a more accurate result (Fig. 8.12).



```
>> t=0:1/60000:1/60;
>> v=311*sin(2*pi*60*t)+100*sin(2*pi*2*60*t)+20*sin(2*pi*3*60*t);
>> rms(v)

ans =

  231.3158

fx >> |
```

FIG. 8.12 Calculation of RMS value of $v(t) = 311\sin(2\pi \times 60t) + 100\sin(2\pi \times 2 \times 60t) + 20\sin(2\pi \times 3 \times 60t)$ with $\frac{1}{60000}$ steps.

8.4.2 Measurement of RMS of signals

The cheap multimeters are not suitable devices to measure the RMS value of signals inside a power electronics converter. The cheap multimeters are able to measure the RMS value of pure sinusoidal signals, i.e., the one shown in Fig. 8.13.

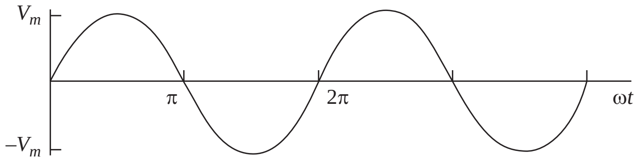


FIG. 8.13 Pure sinusoidal waveform.

Fig. 8.14 shows one of the methods that a cheap multimeter uses to measure the RMS of a signal. V_X is the signal under measurement. Assume that V_X is a pure sinusoidal waveform, i.e., a signal such as the one shown in Fig. 8.13. Then the capacitor is charged up to Vm Volts (voltage drop of diode is neglected), where Vm is the peak value of voltage under measurement. So, Analog-to-Digital converter reads the maximum of input signal. The read value is simply multiplied by $\frac{1}{\sqrt{2}}$, and the result, i.e., $\frac{Vm}{\sqrt{2}}$, is the RMS value of input signal. This method only works for pure sinusoidal signals and doesn't produce correct result if the input signal is not pure sinusoidal.

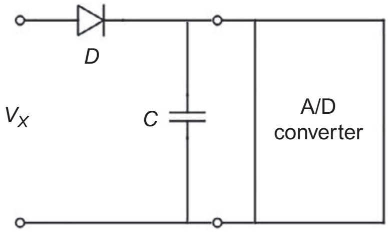


FIG. 8.14 A simple circuit for detection of input AC signal peak value.

The expensive multimeters sample the input waveform and use a processor to calculate the RMS value. So, the waveshape of input signal doesn't affect the measurements. Such a multimeter has "TRUE RMS" label on it. So, ensure that your multimeter is TRUE RMS type if you want to measure the RMS of a signal of a power electronics converters. Digital oscilloscopes can be used to measure the RMS of signals as well.

8.5 Apparent power and power factor

Apparent power (S) is the product of RMS of voltage and RMS of current magnitudes.

$$S = V_{rms} \times I_{rms} \tag{8.15}$$

The power factor of a load is defined as the ratio of average power to apparent power:

$$\text{pf} = \frac{P}{S} = \frac{\text{P}}{\text{V}_{\text{rms}}\text{I}_{\text{rms}}} \tag{8.16}$$

The equation can be used to analyze both the linear circuits and nonlinear circuits. In the linear circuit case, $PF = cos(\alpha)$, where α shows the phase angle between the voltage and current sinusoids.

8.6 Power computations for linear circuits

The steady-state voltages and currents of a linear circuit, which has sinusoidal AC sources, are sinusoidal. Assume an element with the following voltage and current,

$$\begin{aligned} v(t) &= V_m \cos(\omega t + \theta) \\ i(t) &= I_m \cos(\omega t + \varphi) \end{aligned} \tag{8.17}$$

Then the instantaneous power is:

$$p(t) = v(t)i(t) = [V_m \cos(\omega t + \theta)][I_m \cos(\omega t + \varphi)] \tag{8.18}$$

According to basic trigonometric identities:

$$(\cos A)(\cos B) = \frac{1}{2}[\cos(A+B) + \cos(A-B)] \tag{8.19}$$

So, instantaneous power can be written as:

$$p(t) = \left(\frac{V_m I_m}{2}\right)[\cos(2\omega t + \theta + \varphi) + \cos(\theta - \varphi)] \tag{8.20}$$

The average power can be calculated easily:

$$\begin{aligned} p(t) &= \frac{1}{T}\int_0^T p(t)dt = \left(\frac{V_m I_m}{2}\right)\int_0^T [\cos(2\omega t + \theta + \varphi) + \cos(\theta - \varphi)]dt \\ &= \left(\frac{V_m I_m}{2}\right)\cos(\theta - \varphi) = V_{rms} I_{rms} \cos(\theta - \varphi) \end{aligned} \tag{8.21}$$

So, the power factor of circuit is $\frac{V_{rms} I_{rms} \cos(\theta - \varphi)}{V_{rms} I_{rms}} = \cos(\theta - \varphi)$. The average power (measured with units of Watts, W) is the part of power, which is consumed by the resistors in the circuit. In the steady state, no net power is absorbed by an inductor or a capacitor. The term reactive power (measured with units of Volt Ampere Reactive, VAR) is commonly used in conjunction with voltages and currents for inductors and capacitors. Reactive power is characterized by energy storage during one-half of the cycle and energy retrieval during the other half. Reactive power (Q) is calculated as:

$$Q = V_{rms} I_{rms} \sin(\theta - \varphi) \tag{8.22}$$

By convention, inductors absorb positive reactive power and capacitors absorb negative reactive power.

Complex Power (measured with units of Volt Ampere, VA) is defined as $(j = \sqrt{-1})$:

$$\boldsymbol{S} = P + jQ \tag{8.23}$$

Apparent power is the magnitude of complex power:

$$S = |S| = \sqrt{P^2 + Q^2} \tag{8.24}$$

Example 8.5 In the following circuit, $v_1(t) = 311\sin(2 \times \pi \times 50 \times t)$, $L = 0.1\ H$ ve $R = 40\ \Omega$. Determine the apparent power, average (active) power, reactive power, and power factor (Fig. 8.15).

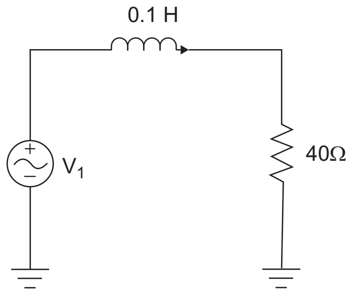


FIG. 8.15 Circuit of Example 8.5.

Solution:

$$\mathrm{Z = R + j \times L \times \omega = 40 + 31.415j}$$

$$\varphi = \tan^{-1}\left(\frac{L\omega}{\mathrm{R}}\right) = \tan^{-1}\left(\frac{31.415}{40}\right) = 38.14° = 0.666\,\mathrm{Rad}$$

$$\mathbf{V} = \frac{311}{\sqrt{2}} < 0° = 219.92\angle 0°$$

$$\mathbf{I} = \frac{\mathrm{V}}{\mathrm{Z}} = \frac{219.92e^{j0}}{40 + 31.42j} = 3.4 - 2.67j = 4.323e^{-0.666j}$$

$$\mathrm{S} = |\mathbf{V} \times \mathbf{I}| = 950.824\,\mathrm{VA}$$

$$\mathrm{P = V \times I \times} \cos(\varphi) = 747.63\,\mathrm{W}$$

$$\mathrm{Q = V \times I \times} \sin(\varphi) = 587.46\,\mathrm{VAR}$$

$$\mathrm{PF} = \cos(\varphi) = 0.786$$

8.7 Fourier series

A periodic and nonsinusoidal signal $f(t)$ that satisfy certain conditions (Dirichlet conditions) can be written as the sum of sinusoids. The Fourier series of $f(t) = f(t+T)$ can be written as ($\omega_0 = \frac{2\pi}{T}$):

$$f(t) = a_0 + \sum_{n=1}^{\infty} [a_n \cos(n\omega_0 t) + b_n \sin(n\omega_0 t)] \tag{8.25}$$

where a_0, a_n, and b_n are

$$\begin{aligned} a_0 &= \frac{1}{T}\int_{-\frac{T}{2}}^{\frac{T}{2}} f(t)dt \\ a_n &= \frac{2}{T}\int_{-\frac{T}{2}}^{\frac{T}{2}} f(t)\cos(n\omega_0 t)dt \\ b_n &= \frac{2}{T}\int_{-\frac{T}{2}}^{\frac{T}{2}} f(t)\sin(n\omega_0 t)dt \end{aligned} \tag{8.26}$$

The $a_0 = \frac{1}{T}\int_{-\frac{T}{2}}^{\frac{T}{2}} f(t)dt$ is called the average value of $f(t)$. The equations can be written in the following forms as well (remember that $a \times \sin(\omega t) + b \times \cos(\omega t) = \sqrt{a^2 + b^2}\sin\left(\omega t + \tan^{-1}\left(\frac{b}{a}\right)\right)$).

(A) Sum of Sines

$$\begin{gathered} f(t) = a_0 + \sum_{n=1}^{\infty} C_n \sin(n\omega_0 t + \theta_n) \\ C_n = \sqrt{a_n{}^2 + b_n{}^2} \text{ and } \theta_n = \tan^{-1}\left(\frac{a_n}{b_n}\right) \end{gathered} \tag{8.27}$$

(B) Sum of Cosines

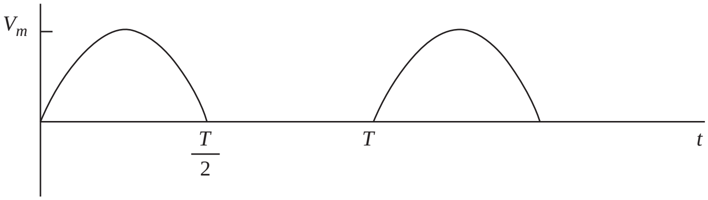
$$\begin{gathered} f(t) = a_0 + \sum_{n=1}^{\infty} C_n \cos(n\omega_0 t + \theta_n) \\ C_n = \sqrt{a_n{}^2 + b_n{}^2} \text{ and } \theta_n = \tan^{-1}\left(-\frac{b_n}{a_n}\right) \end{gathered} \tag{8.28}$$

The following equation can be used to determine the RMS value of a signal using its Fourier series coefficients.

$$F_{rms} = \sqrt{\sum_{n=0}^{\infty} F_{n,rms}{}^2} = \sqrt{a_0{}^2 + \sum_{n=1}^{\infty}\left(\frac{C_n}{\sqrt{2}}\right)^2} = \sqrt{a_0{}^2 + \sum_{n=1}^{\infty}\left(\frac{a_n{}^2 + b_n{}^2}{2}\right)} \tag{8.29}$$

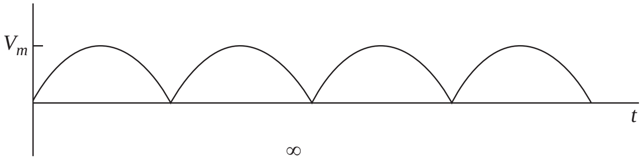
8.7.1 Fourier series of important waveshapes

Fourier series of important waveshapes are shown in Figs. 8.16–8.20.



$$v(t) = \frac{V_m}{\pi} + \frac{V_m}{2}\sin(\omega_0 t) - \sum_{n=2,4,6...}^{\infty} \frac{2V_m}{(n^2-1)\pi}\cos(n\omega_0 t)$$

FIG. 8.16 Fourier series of a half-wave rectified waveform.

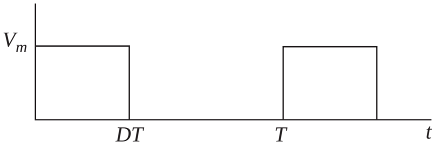


$$v_o(t) = V_o + \sum_{n=2,4,...}^{\infty} V_n \cos(n\omega_0 t + \pi)$$

$$V_o = \frac{2V_m}{\pi}$$

$$V_n = \frac{2V_m}{\pi}\left(\frac{1}{n-1} - \frac{1}{n+1}\right)$$

FIG. 8.17 Fourier series of a full-wave rectified waveform.



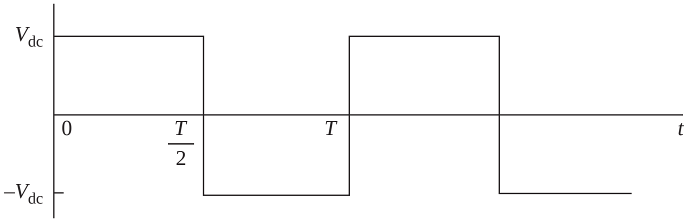
$$v_o(t) = a_0 + \sum_{n=1}^{\infty}[a_n \cos(n\omega_0 t) + b_n \sin(n\omega_0 t)]$$

$$a_0 = V_m D$$

$$a_n = \left(\frac{V_m}{n\pi}\right)\sin(n2\pi D)$$

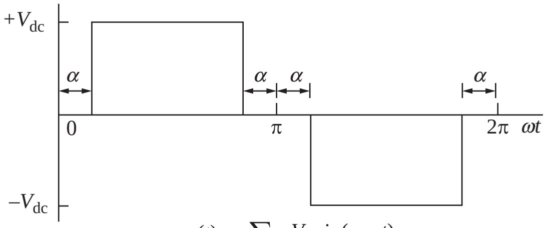
$$b_n = \left(\frac{V_m}{n\pi}\right)[1 - \cos(n2\pi D)]$$

FIG. 8.18 Fourier series of a pulsed waveform.



$$v_o(t) = \sum_{n=1,3,5,\ldots} \left(\frac{4V_{dc}}{n\pi}\right) \sin(n\omega_0 t)$$

FIG. 8.19 Fourier series of a square wave.



$$v_o(t) = \sum_{n=1,3,5,\ldots} V_n \sin(n\omega_0 t)$$

$$V_n = \left(\frac{4V_{dc}}{n\pi}\right) \cos(n\alpha)$$

FIG. 8.20 Fourier series of a modified square wave.

8.7.2 Calculation of average power using the Fourier series

Assume that the Fourier series of voltage and current of a element is given as follows

$$\begin{aligned} v(t) &= V_0 + \sum_{n=1}^{\infty} V_n \cos(n\omega_0 t + \theta_n) \\ i(t) &= I_0 + \sum_{n=1}^{\infty} I_n \cos(n\omega_0 t + \varphi_n) \end{aligned} \tag{8.30}$$

Then the average power (i.e., $\frac{1}{T}\int_0^T v(t)i(t)dt$)) can be calculated as:

$$P = \sum_{n=0}^{\infty} P_n = V_0 I_0 + \sum_{n=1}^{\infty} V_{n,rms} I_{n,rms} \cos(\theta_n - \varphi_n) \tag{8.31}$$

or

$$P = V_0 I_0 + \sum_{n=1}^{\infty} \frac{V_{n,\max} I_{n,\max}}{2} \cos(\theta_n - \varphi_n) \tag{8.32}$$

For instance, the average power for $v(t)=10+20\cos(2\pi\times 60t)+30\cos(4\pi\times 60t+30°)$ and $i(t)=2+2.65\cos(2\pi\times 60t-48.5°)+2.43\cos(4\pi\times 60t-36.2°)$ is 52.2 W.

8.8 Total harmonic distortion (THD)

THD quantifies the nonsinusoidal property of a waveform. THD is often applied in situations where the dc term is zero. Assume that the Fourier series of the signal is given ($f(t)$ can be either voltage or current waveform):

$$f(t)=\sum_{n=1}^{\infty}\left[a_n\cos\left(n\omega_0 t\right)+b_n\sin\left(n\omega_0 t\right)\right] \tag{8.33}$$

Then the THD of signal is defined as:

$$\mathrm{THD}=\sqrt{\frac{{F_{rms}}^2-{F_{1,rms}}^2}{{F_{1,rms}}^2}}, \tag{8.34}$$

where F_{rms} and $F_{1,\ rms}$ show the RMS value of signal $f(t)$ and RMS value of fundamental harmonic of $f(t)$ (note that $F_{rms}=\sqrt{\sum_{n=1}^{\infty}\frac{\left(a_n{}^2+b_n{}^2\right)}{2}}$ and $F_{1,rms}=\sqrt{\frac{\left(a_1{}^2+b_1{}^2\right)}{2}}$). For instance for a current waveform of $i(t)=4\sin(\omega_0 t)+1.5\sin(3\omega_0 t)+0.64\sin(5\omega_0 t)$, the THD is $\sqrt{\frac{3.0545^2-2.829^2}{2.829^2}}=0.408$. It is quite common to express the THD in percentage, so the THD for aforementioned current waveform is 40.8%.

Example 8.6 Determine the THD of $v(t)=311\sin(2\pi\times 60t)+100\sin(2\pi\times 2\times 60t)+20\sin(2\pi\times 3\times 60t)$.

Solution

RMS of given waveforms is $V_{rms}=\sqrt{\left(\frac{311}{\sqrt{2}}\right)^2+\left(\frac{100}{\sqrt{2}}\right)^2+\left(\frac{20}{\sqrt{2}}\right)^2}=231.43\,\mathrm{V}$. Peak value of fundamental harmonic is 311 V. So, the RMS value of fundamental harmonic is $V_{1,rms}=\frac{311}{\sqrt{2}}=219.91\,\mathrm{V}$. Finally the THD is:

$$\mathrm{THD}=\sqrt{\frac{{V_{rms}}^2-{V_{1,rms}}^2}{{V_{1,rms}}^2}}=\sqrt{\frac{231.43^2-219.91^2}{219.91^2}}=0.33 \text{ or } 33\%$$

Example 8.7 Determine the THD for the given voltage waveform

$$v(t)=\begin{cases}-100 & -1ms<t<0\\ +100 & 0<t<1ms\end{cases}$$

Solution

The graph of one period of given waveform is shown in Fig. 8.21.

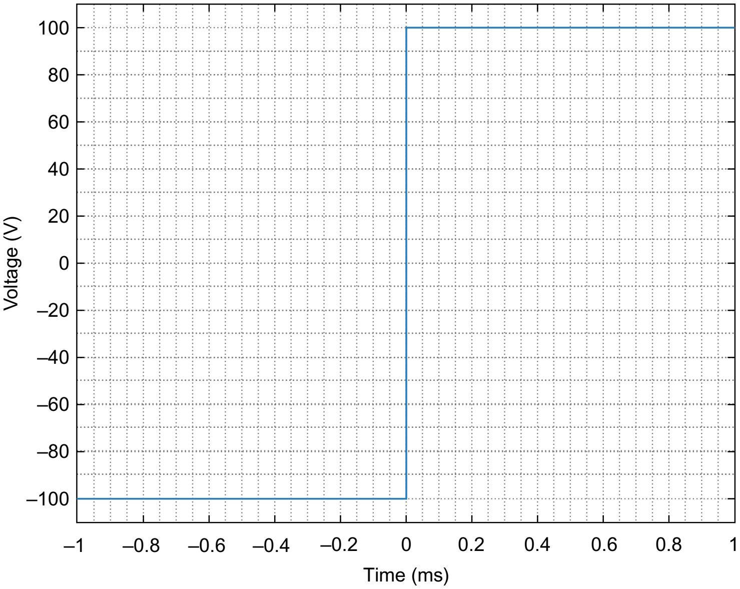


FIG. 8.21 Graph of given $v(t)$.

Fourier series of $v(t)$ is:

$$v(t) = \sum_{n=1,3,5}^{\infty} \frac{2 \times 200}{n\pi} \sin(n\omega_0 t) \tag{8.35}$$

RMS value of $v(t)$ is:

$$V_{rms} = \sqrt{\frac{1}{T}\int_0^T v(t)^2 dt} = \sqrt{\frac{1}{2m}\int_{-1m}^{0} (-100)^2 dt + \frac{1}{2m}\int_0^{1m} (100)^2 dt} = 100\,V \tag{8.36}$$

Fundamental harmonic (first harmonic) of $v(t)$ has the peak value of $V_1 = \frac{2\times 200}{\pi} = 127.324\,V$. So the RMS value of fundamental harmonic is $V_{1,rms} = \frac{V_1}{\sqrt{2}} = \frac{127.324}{\sqrt{2}} = 90.0325\,V$. Finally the THD is:

$$\text{THD} = \sqrt{\frac{{V_{rms}}^2 - {V_{1,rms}}^2}{{V_{1,rms}}^2}} = \sqrt{\frac{100^2 - 90.0325^2}{90.0325^2}} = 0.4834 \text{ or } 48.34\% \tag{8.37}$$

8.9 Volt-second balance

Volt-Second balance is one of the important tools in analyzing the power electronics converters (specially in analyzing the DC-DC converters).

The voltages and currents of a power electronics converter are periodic, i.e., $i(t+T)=i(t)$ and $v(t+T)=v(t)$. The relation between the voltage and current of an inductor is:

$$v_L(t)=L\frac{di_L(t)}{dt} \tag{8.38}$$

So,

$$i(t_0+T)=\frac{1}{L}\int_{t_0}^{t_0+T} v_L(t)dt+i(t_0) \Longrightarrow i(t_0+T)-i(t_0)=\frac{1}{L}\int_{t_0}^{t_0+T} v_L(t)dt \tag{8.39}$$

Since the current is assumed to be periodic, the starting and end values are the same, i.e., $i(t_0+T)-i(t_0)=0$. So,

$$i(t_0+T)-i(t_0)=\frac{1}{L}\int_{t_0}^{t_0+T} v_L(t)dt=0 \tag{8.40}$$

Both sides are multiplied with $\frac{L}{T}$. So,

$$\text{avg}[v_L(t)]=V_L=\frac{1}{T}\int_{t_0}^{t_0+T} v_L(t)dt=0 \tag{8.41}$$

Therefore, for periodic currents, the average voltage across an inductor is zero. This is called Volt-Second Balance.

8.10 Ampere-second balance

Dual of previous law can be extracted for capacitors. The relation between the voltage and current of a capacitor is:

$$i_C(t)=C\frac{dv_C(t)}{dt} \tag{8.42}$$

So,

$$v(t_0+T)=\frac{1}{C}\int_{t_0}^{t_0+T} i_C(t)dt+v(t_0) \Longrightarrow v(t_0+T)-v(t_0)=\frac{1}{C}\int_{t_0}^{t_0+T} i_C(t)dt \tag{8.43}$$

Since the voltage is assumed to be periodic, the starting and end values are the same, i.e., $v(t_0+T)-v(t_0)=0$. So,

$$v(t_0+T)-v(t_0)=\frac{1}{C}\int_{t_0}^{t_0+T} i_C(t)dt=0 \tag{8.44}$$

Both sides are multiplied with $\frac{C}{T}$. So,

$$\text{avg}[i_C(t)]=I_C=\frac{1}{T}\int_{t_0}^{t_0+T} i_C(t)dt=0 \tag{8.45}$$

Therefore, for periodic voltages, the average current in a capacitor is zero. This is called Ampere-Second Balance.

8.11 MOSFET with resistive load

Consider a MOSFET with a resistive load.

The MOSFET is voltage-controlled switch, i.e., its on/off status is determined with gate voltage. When the gate-source voltage is greater than the threshold voltage, the MOSFET acts as a closed switch; otherwise, the device acts as an open switch.

Assume that the MOSFET M in Fig. 8.22 is supplied with a periodic gate-source voltage with duty ratio D. The amplitude of periodic gate-source voltage is large enough to close the MOSFET. Fig. 8.23 shows the circuit waveforms.

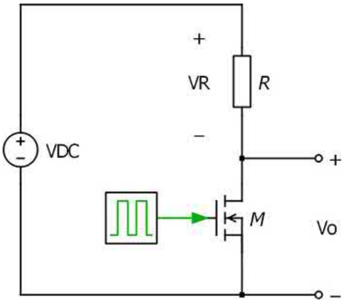


FIG. 8.22 A MOSFET with purely resistive load.

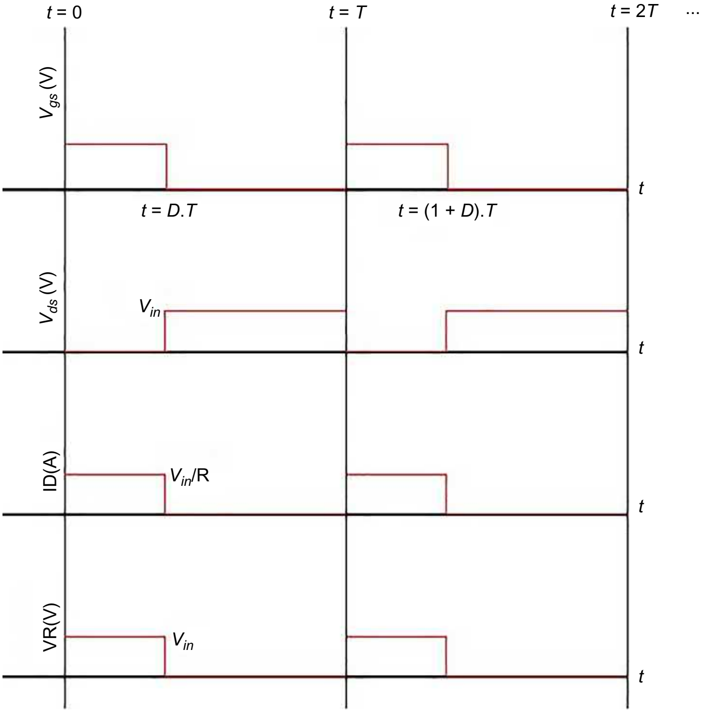


FIG. 8.23 V_{gs}, V_{ds}, ID, and VR show the gate-source voltage, drain-source voltage, drain current, and load resistor voltage, respectively. The MOSFET is assumed to be ideal, i.e., drain source resistance is ignored.

The RMS and average of waveforms can be calculated easily using the formulas provided before. For instance, the average and RMS value of load voltage is D. Vin and $\sqrt{D}$.Vin, respectively. So, the average power dissipated in the resistor is $\frac{(Vrms)^2}{R} = \frac{\left(\sqrt{D}.\mathrm{Vin}\right)^2}{R} = \frac{D.\mathrm{Vin}^2}{R}$.

The scheme used in Fig. 8.22 is called low-side switching. Note that the source of MOSFET is connected to the ground. The scheme used in Fig. 8.24 is called high-side switching. Note that the source of MOSFET is NOT connected to the ground.

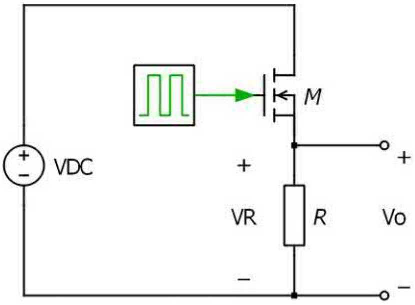


FIG. 8.24 High-side switching.

The two schemes are the same from load point of view. However, triggering a low-side MOSFET is quite easier than a high-side one. Studying a numeric example is quite helpful for this purpose. Assume that R = 10 Ω, VDC = 20 V,$r_{ds,on} = 20\ m\Omega$, and $V_{TH} = 8\ V$. $r_{ds,on}$ and V_{TH} show the drain-source resistance for a closed MOSFET and threshold voltage of MOSFET, respectively. The gate-source voltage must exceed the threshold voltage in order to turn on the MOSFET. In Fig. 8.22, the source of MOSFET is connected to ground. So, $V_S = 0$. The MOSFET in Fig. 8.22 is closed if

$$V_{GS} > V_{TH} \Rightarrow V_G - V_S > V_{TH} \Rightarrow V_G - 0 > V_{TH} \Rightarrow V_G > V_{TH} \tag{8.46}$$

So, the MOSFET in Fig. 8.22 will be closed if a voltage greater than 8 V is applied to its gate.

When the high-side MOSFET of Fig. 8.24 is closed, the source voltage is:

$$\mathrm{V_S} = \frac{\mathrm{R}}{\mathrm{R} + r_{ds,on}} \times \mathrm{VDC} = \frac{10}{10 + 0.02} \times 20 = 19.96\,V \tag{8.47}$$

So, in order to keep the device close, the gate voltage must satisfy the following in equality:

$$V_{GS} > V_{TH} \Rightarrow V_G - V_S > V_{TH} \Rightarrow V_G - 19.96 > V_{TH} \Rightarrow V_G > 19.96 + 8 \Rightarrow V_G > 27.96\,V \tag{8.48}$$

So, in order to close the high-side MOSFET of Fig. 8.24, a voltage larger than supply voltage is required. High-side MOSFETs can be triggered with the aid of a technique called bootstrapping. In this technique, the electric charge is saved in a capacitor and the capacitor is used to trigger the MOSFET in the required time instant. So, there is no need for a voltage bigger than the supply voltage in order to turn on the MOSFET. The IR2101 IC uses the bootstrapping technique to trigger a high-side switch.

8.12 Uncontrolled half-wave rectifier

A rectifier is a circuit, which converts the AC input into a DC output. Fig. 8.25 shows the simplest rectifier circuit. It consists of an AC source, a diode, and a load.

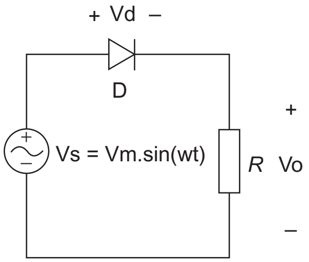


FIG. 8.25 Simple half-wave rectifier. The output voltage is not controllable.

The waveforms of the circuit are shown in Fig. 8.26. The voltage drop of diode can be ignored if the peak value of input AC source is much bigger than the forward voltage drop of diode.

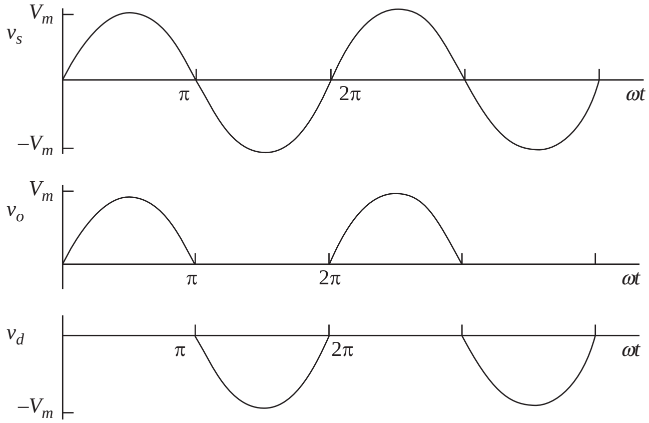


FIG. 8.26 Typical waveforms of circuit shown in Fig. 8.25. Addition of a capacitor in parallel with the output load decreases the harmonics in load voltage and leads to a more smooth output voltage.

Average value of load voltage is:

$$V_O = V_{avg} = \frac{1}{2\pi}\int_0^{\pi} V_m \sin(\omega t) d(\omega t) = \frac{V_m}{\pi} \tag{8.49}$$

The average value of load current is:

$$I_O = \frac{V_O}{R} = \frac{V_m}{\pi R} \tag{8.50}$$

The average power dissipated in resistor is $P = I^2{}_{rms}R = \frac{V_{rms}{}^2}{R}$. The RMS value of load voltage and current is:

$$\begin{aligned} V_{rms} &= \sqrt{\frac{1}{2\pi}\int_0^{\pi}[V_m \sin(\omega t)]^2 d(\omega t)} = \frac{V_m}{2} \\ I_{rms} &= \frac{V_{rms}}{R} = \frac{V_m}{2R} \end{aligned} \tag{8.51}$$

For instance for $v_s(t) = 120\sqrt{2}\sin(2\pi \times 60 \times t)\,V$ and $R = 5\ \Omega$, $I_{load,avg} = 10.8$ A, $I_{load,rms} = 17$ A, $V_{avg} = 54$ V, $V_{rms} = 84.9$ V and $P = 1440$ W.

8.13 Controlled half-wave rectifier

The output voltage of half-wave diode rectifier is not controllable (Fig. 8.27). The output voltage can be controlled if the diode is replaced with a thyristor. The thyristor requires gate signal in order to turn on. Only the turn-on instant is under control. The turn-off instant is not under control; it is governed by the circuit.

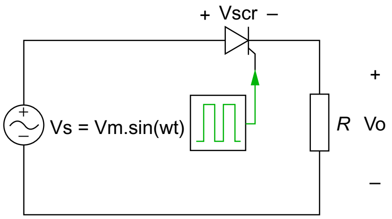


FIG. 8.27 A simple half-wave rectifier with controllable output voltage.

Fig. 8.28 shows the typical waveforms of circuit shown in Fig. 8.27.

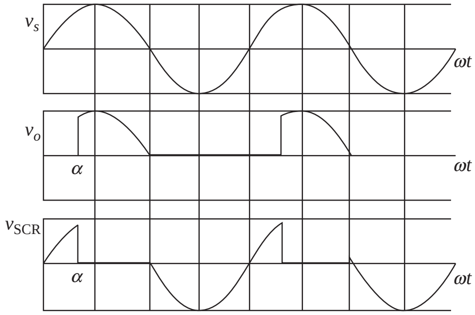


FIG. 8.28 Typical waveforms of circuit shown in Fig. 8.27.

The thyristor is fired at $\omega t = \alpha$. The thyristor is closed for $\alpha < \omega t < \pi$. The thyristor is off during the $0 < \omega t < \alpha$ (since the gate signal is not applied yet) and $\pi < \omega t < 2\pi$ (since the thyristor is a unidirectional switch, i.e., the current only flows from anode toward the cathode)

The average and RMS value of load voltage can be calculated using the following formulas:

$$V_O = V_{avg} = \frac{1}{2\pi}\int_{\alpha}^{\pi} V_m \sin(\omega t) d(\omega t) = \frac{V_m}{2\pi}(1 + \cos\alpha)$$

$$V_{rms} = \sqrt{\frac{1}{2\pi}\int_{0}^{2\pi} v_o(\omega t)^2 d(\omega t)} = \sqrt{\frac{1}{2\pi}\int_{\alpha}^{\pi} [V_m \sin(\omega t)]^2 d(\omega t)}$$
$$= \frac{V_m}{2}\sqrt{1 - \frac{\alpha}{\pi} + \frac{\sin(2\alpha)}{2\pi}} \tag{8.52}$$

If we substitute $\alpha = 0$ in the equations, we obtain the formulas for uncontrolled case (Fig. 8.25). The average value and RMS value of output voltage decrease with increasing firing angle (α). The average power dissipated in the resistor can be calculated using the $P_R = \frac{{V_{rms}}^2}{R}$.

For instance for $v_s(t) = 120\sqrt{2}\sin(2\pi \times 60 \times t)\,V$, $R = 100\ \Omega$ and $\alpha = 61.2°$, $I_{load,\ avg} = 0.4\,A$, $I_{load,\ rms} = 0.76$ A, $V_{avg} = 40$ V, $V_{rms} = 75.6$ V,and $P = 57.1$ W.

8.14 DC-DC converters

A DC-DC converter, as the name suggests, converts a DC voltage to another DC voltage. The output voltage can be greater than or less than input voltage. The DC-DC converters can be divided into two broad categories:

1. Isolated converters,
2. Nonisolated converters.

The isolated converters contain high-frequency transformers (i.e., a transformer with Ferrite core). So, there is an isolation between the source supply the converter and the load. Flyback, forward or half/full bridge converters are examples of isolated converters.

The nonisolated converters do not contain high-frequency transformers. So, there is no isolation between the input DC source and load. Buck, boost, buck-boost, Cuk, and zeta are examples of nonisolated converters.

The DC-DC converters can operate in two modes: CCM (Continuous Current Mode) and DCM (Discontinuous Current Mode). The operating mode of converter is determined based on the steady-state waveform of inductor current. If the minimum of steady-state inductor current is larger than zero, then the converter operates in CCM (Fig. 8.29).

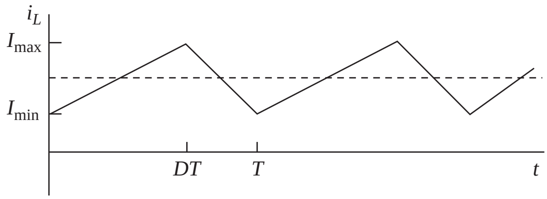


FIG. 8.29 Typical inductor waveform for a converter operating in CCM. Steady-state minimum value of inductor current is positive.

If steady-state inductor current is zero for some time during the switching period, then the converter operates in DCM (Fig. 8.30).

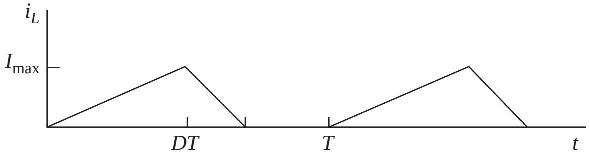


FIG. 8.30 Typical inductor waveform for a converter operating in CCM.

8.14.1 Buck converter

Buck converter decreases the input voltage, i.e., $V_O < V_{IN}$. Fig. 8.31 shows the schematic of a buck converter.

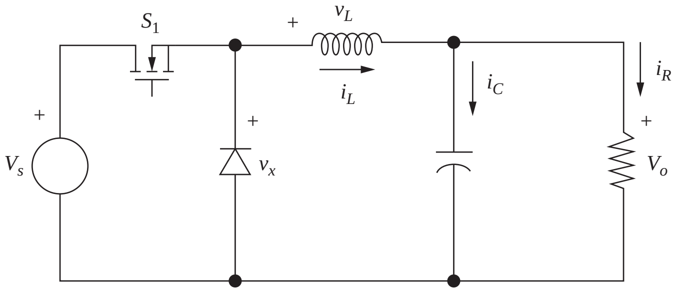


FIG. 8.31 Schematic of buck converter.

The buck converter is analyzed under the following assumptions:

1. The circuit reaches its steady state, i.e., the transient has been finished.
2. The converter operates in CCM, i.e., minimum of inductor current in steady state is positive.
3. The capacitor is large enough so the output voltage is held constant at V_O, i.e., the output ripple is negligible.

4. The switching period is T and the MOSFET is closed for $D.\ T$ interval. $0 < D < 1$ is duty ratio of control signal applied to gate of MOSFET.
5. The components are ideal, i.e., drain-source resistance, forward voltage drop of diode, ESR of capacitor/inductor are zero.

When the MOSFET switch is closed, the diode is revere biased and acts as an open circuit. The equivalent circuit shown in Fig. 8.32 represents this case.

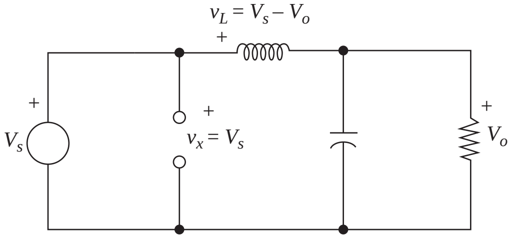


FIG. 8.32 Equivalent circuit of buck converter for closed MOSFET. MOSFET remains closed for duration of $D \times T$ seconds.

When the MOSFET switch is opened, the diode is forward biased and acts as a short circuit. The equivalent circuit shown in Fig. 8.33 represents this case.

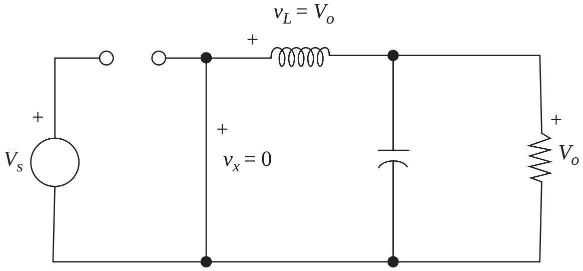


FIG. 8.33 Equivalent circuit of buck converter for opened MOSFET. MOSFET remains open for duration of $(1 - D) \times T$ seconds.

8.14.1.1 Calculation of output voltage

According to Fig. 8.32, the inductor voltage (for an interval of length $D.\ T$) is:

$$v_L = V_s - V_o \tag{8.53}$$

According to Fig. 8.33, the inductor voltage (for an interval of length $(1 - D).\ T$) is:

$$v_L = -V_o \tag{8.54}$$

Fig. 8.34 shows the voltage of inductor.

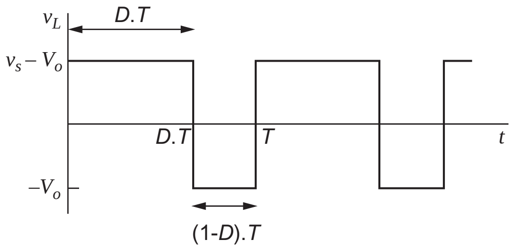


FIG. 8.34 Waveform of inductor voltage.

The Volt-Second balance can be used to calculate the unknown value of output voltage:

$$(V_s - V_o) \times D \times T - V_o \times (1 - D) \times T = 0 \Rightarrow V_o = D \times V_s \tag{8.55}$$

The output voltage is less than the input voltage (remember that $0 < D < 1$).

The same result can be obtained within a different approach: During the interval, which MOSFET is closed (Fig. 8.32), the inductor voltage can be written as:

$$\begin{aligned} v_L &= V_s - V_o = L\frac{d}{dt}i_L(t) \\ \frac{d}{dt}i_L(t) &= \frac{V_s - V_o}{L} \end{aligned} \tag{8.56}$$

Since the V_o is less than V_s, $\frac{V_s - V_o}{L}$ is a positive value and the inductor current increases during the interval MOSFET is closed. The increase in the inductor current during this interval is:

$$\begin{aligned} \frac{d}{dt}i_L(t) &= \frac{\Delta i_L}{\Delta t} = \frac{\Delta i_L}{D \times T} = \frac{V_s - V_O}{L} \\ \Delta i_{L,\,\text{Closed MOSFET}} &= \frac{V_s - V_O}{L} \times D \times T \end{aligned} \tag{8.57}$$

During the interval which MOSFET is opened (Fig. 8.33), the inductor voltage can be written as:

$$\begin{aligned} v_L &= -V_o = L\frac{d}{dt}i_L(t) \\ \frac{d}{dt}i_L(t) &= \frac{-V_o}{L} \end{aligned} \tag{8.58}$$

Since $\frac{-V_o}{L}$ is a negative value, the inductor current decreases during this interval. The amount of decrease in the inductor current during this interval is:

$$\begin{aligned} \frac{d}{dt}i_L(t) &= \frac{\Delta i_L}{\Delta t} = \frac{\Delta i_L}{(1 - D) \times T} = \frac{-V_O}{L} \\ \Delta i_{L,\text{Opened MOSFET}} &= \frac{-V_O}{L} \times (1 - D) \times T \end{aligned} \tag{8.59}$$

Since the inductor current is periodic, i.e., $i_L(t+T)=i_L(t)$, the following equation can be written:

$$\begin{aligned}&\Delta i_{L,\text{Closed MOSFET}}+\Delta i_{L,\text{Opened MOSFET}}=0\\&\frac{V_s-V_O}{L}\times D\times T+\frac{-V_O}{L}\times(1-D)\times T=0\end{aligned}\tag{8.60}$$

which leads to

$$V_O=D\times V_s\tag{8.61}$$

8.14.1.2 Calculation of average current drawn from input source

The average input power of converter is $P_{in}=V_s\times I_s$, where I_s is the average current drawn from the input DC source. Since we assumed that converter components are ideal, the efficiency is 100%, i.e., all the input power is dissipated in the output load. So,

$$\begin{aligned}&P_{in}=P_{out}\\&V_s\times I_s=V_o\times I_o\\&I_s=\frac{V_o\times I_o}{V_s}=\frac{V_o{}^2}{R\times V_s}=\frac{(D\times V_s)^2}{R\times V_s}=\frac{D^2\times V_s}{R}\end{aligned}\tag{8.62}$$

8.14.1.3 Determining the operating mode of converter

In this section, we want to determine under which condition the converter operates in CCM. Maximum and minimum of inductor current can be written as $I_{max}=I_L+\frac{\Delta i_L}{2}$ and $I_{min}=I_L-\frac{\Delta i_L}{2}$. I_L is the average value of inductor current (Fig. 8.35).

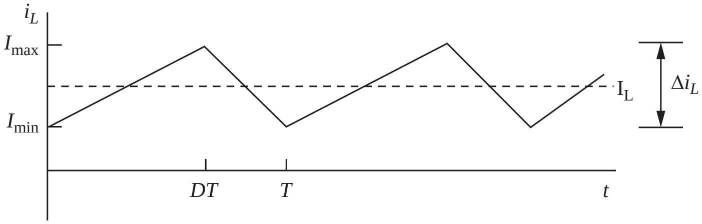


FIG. 8.35 Maximum and minimum of inductor current is $I_{max}=I_L+\frac{\Delta i_L}{2}$ and $I_{min}=I_L-\frac{\Delta i_L}{2}$. The I_L and $\triangle i_L$ show the average value of inductor current and ripple of inductor current, respectivley.

Value of $\triangle i_L$ is calculated in the previous analysis. Its value can be calcultated using one of the following formulas:

$$\begin{aligned}&\Delta i_L=\Delta i_{L,\text{Opened MOSFET}}=\frac{-V_O}{L}\times(1-D)\times T\\&\Delta i_L=\Delta i_{L,\text{Closed MOSFET}}=\frac{V_s-V_O}{L}\times D\times T\end{aligned}\tag{8.63}$$

We need to determine the value of I_L in order to find the value of I_{max} and I_{min}. The value of of I_L can be found with the aid of Fig. 8.31. According to KCL (Kirchhoff's Current Law),

$$i_L = i_C + i_R \tag{8.64}$$

We calculate the average $(\frac{1}{T}\int_{t_0}^{t_0+T} f(\tau)d\tau)$ of both sides:

$$\frac{1}{T}\int_{t_0}^{t_0+T} i_L(\tau)d\tau = \frac{1}{T}\int_{t_0}^{t_0+T} i_C(\tau)d\tau + \frac{1}{T}\int_{t_0}^{t_0+T} i_R(\tau)d\tau \tag{8.65}$$

The average value of capacitor voltage is zero (remember the Ampere-Second Balance). So,

$$\frac{1}{T}\int_{t_0}^{t_0+T} i_L(\tau)d\tau = \frac{1}{T}\int_{t_0}^{t_0+T} i_R(\tau)d\tau \tag{8.66}$$

The average value of resistor current can be found easily using Ohm's law

$$\frac{1}{T}\int_{t_0}^{t_0+T} i_R(\tau)d\tau = \frac{V_O}{R} = \frac{D.V_s}{R} \tag{8.67}$$

So, the average value of inductor current is:

$$I_L = \frac{1}{T}\int_{t_0}^{t_0+T} i_L(\tau)d\tau = \frac{V_O}{R} = \frac{D.V_s}{R} \tag{8.68}$$

Once $I_L = \frac{V_O}{R}$ and $\Delta i_L = \left|\Delta i_{L,\text{Opened MOSFET}}\right| = \frac{V_O}{L} \times (1-D) \times T$ are substituted in the $I_{max} = I_L + \frac{\Delta i_L}{2}$ and $I_{min} = I_L - \frac{\Delta i_L}{2}$ equations,

$$\begin{aligned} I_{max} &= I_L + \frac{\Delta i_L}{2} = \frac{V_o}{R} + \frac{1}{2}\left[\frac{V_o}{L}(1-D)T\right] = V_o\left(\frac{1}{R} + \frac{1-D}{2Lf}\right) \\ I_{min} &= I_L - \frac{\Delta i_L}{2} = \frac{V_o}{R} - \frac{1}{2}\left[\frac{V_o}{L}(1-D)T\right] = V_o\left(\frac{1}{R} - \frac{1-D}{2Lf}\right) \end{aligned} \tag{8.69}$$

are obtained. f is the switching frequency and $f = \frac{1}{T}$.

The buck converter operates in CCM if, $I_{min} > 0$. This inequality leads to:

$$L > L_{min} = \frac{(1-D) \times R}{2f} \tag{8.70}$$

So, if the buck converter bobbin is larger than the minimum value, then the converter operates in CCM; otherwise, it operates in DCM.

8.14.1.4 Calculation of output ripple

In the previous analysis, the capacitor was assumed to be very large to keep the output voltage constant. In practice, the output voltage cannot be kept perfectly constant with a finite capacitance. Effect of finite capacitance on output voltage variations (ripple) is studied in this section.

According to Fig. 8.31, the current in capacitor is

$$i_C = i_L - i_R \tag{8.71}$$

The inductor and capacitor currents are shown in Fig. 8.36.

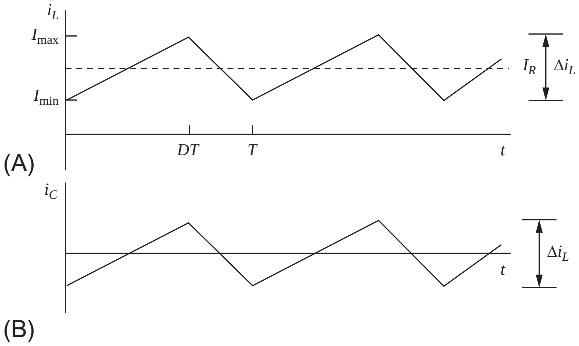


FIG. 8.36 (A) Inductor current, (B) capacitor current.

The relation between output voltage ripple ($\triangle V_O$) and change in capacitor charge ($\triangle Q$) is

$$\triangle Q = C \times \triangle V_O \Longrightarrow \triangle V_O = \frac{\triangle Q}{C} \tag{8.72}$$

The change in capacitor charge can be calculated easily using Fig. 8.37.

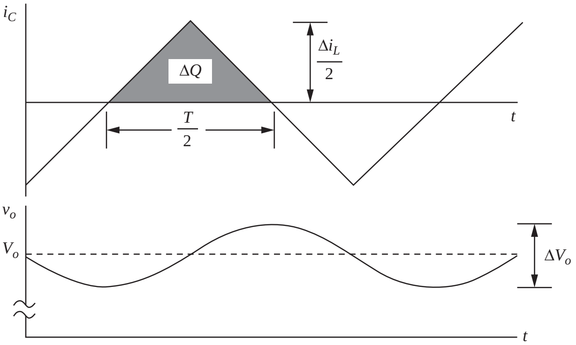


FIG. 8.37 Waveforms of capacitor current and output voltage of buck converter.

According to Fig. 8.37,

$$\triangle Q = \frac{1}{2} \times \frac{T}{2} \times \frac{\triangle i_L}{2} = \frac{T \times \triangle i_L}{8} \tag{8.73}$$

So,

$$\triangle V_O = \frac{\triangle Q}{C} = \frac{T \times \triangle i_L}{8 \times C} \tag{8.74}$$

Since $\triangle i_L = \frac{V_o}{L} \times (1-D) \times T$,

$$\triangle V_O = \frac{T \times \triangle i_L}{8 \times C} = \frac{T \times V_o}{8 \times C \times L}(1-D)T = \frac{V_o(1-D)}{8LCf^2} \Rightarrow \frac{\triangle V_O}{V_o} = \frac{1-D}{8LCf^2} \tag{8.75}$$

In design, it is useful to rearrange the preceding equation to express required capacitance in terms of specified voltage ripple:

$$C = \frac{1-D}{8L\left(\frac{\triangle V_O}{V_o}\right)f^2} \tag{8.76}$$

For instance, for $V_s = 50$ V, $L = 400$ μH, $R = 20$ Ω, $C = 100$ μF, $f = 10$ kHz, and $D = 0.4$, the converter operates in CCM since $L = 400\,\mu\text{H} > L_{min} = \frac{(1-D)\times R}{2f} = \frac{(1-0.4)\times 20}{2\times 20000} = 300\,\mu\text{H}$, the output voltage is 20 V, $I_{max} = 1.75$ A, $I_{min} = 0.25$ A, and output voltage ripple is 94 mV.

8.15 Calculation of output voltage of a buck converter operated in DCM

In this section, we show how the output voltage a buck converter operating in DCM can be calculated. Assume that the buck converter shown in Fig. 8.31 is operated in DCM. The converter waveforms are shown in Fig. 8.38.

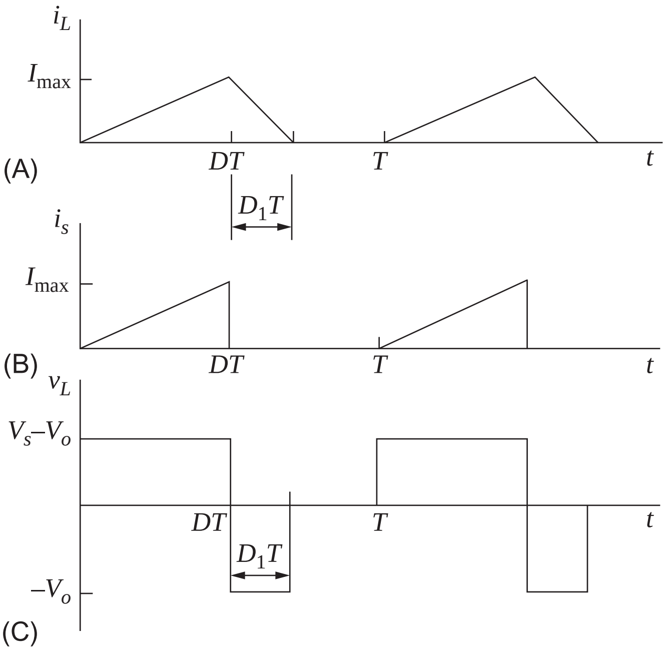


FIG. 8.38 Different waveforms for a buck converter operating in DCM. i_L, i_s, and v_L show the inductor current, the current drawn from input DC source, and inductor voltage, respectively.

According to Volt-Second Balance

$$\begin{aligned}&(V_s - V_o)DT - V_oD_1T = 0\\&(V_s - V_o)D = V_oD_1\end{aligned} \tag{8.77}$$

or

$$\frac{V_o}{V_s} = \left(\frac{D}{D+D_1}\right) \tag{8.78}$$

As explained in Section 8.9.3.4, the average value of inductor current (I_L) is the same as average value of load current (I_R)

$$I_L = I_R = \frac{V_o}{R} \tag{8.79}$$

Average value of resistor current can be found using the $I_R = \frac{V_O}{R}$, where V_O is the average value of output voltage. According to Fig. 8.38A, the average value of inductor current is:

$$I_L = \frac{1}{T}\left(\frac{1}{2}I_{max}DT + \frac{1}{2}I_{max}D_1T\right) = \frac{1}{2}I_{max}(D+D_1) \tag{8.80}$$

So, $\frac{1}{2}I_{max}(D+D_1) = \frac{V_O}{R}$. When the MOSFET is closed, the inductor current starts increasing from zero. During the interval MOSFET is closed, the inductor voltage is:

$$v_L = V_s - V_o \tag{8.81}$$

So,

$$\frac{di_L}{dt} = \frac{V_s - V_o}{L} = \frac{\Delta i_L}{\Delta t} = \frac{\Delta i_L}{DT} = \frac{I_{max}}{DT} \tag{8.82}$$

Since $V_S = \frac{D+D_1}{D}V_O$,

$$I_{max} = \Delta i_L = \left(\frac{V_s - V_o}{L}\right)DT = \frac{V_oD_1T}{L} \tag{8.83}$$

If we substitute the $I_{max} = \frac{V_OD_1T}{L}$ into the $\frac{1}{2}I_{max}(D+D_1) = \frac{V_O}{R}$, we obtain:

$$\begin{aligned}&\frac{1}{2}I_{max}(D+D_1) = \frac{1}{2}\left(\frac{V_oD_1T}{L}\right)(D+D_1) = \frac{V_o}{R}\\&D_1{}^2 + DD_1 - \frac{2L}{RT} = 0\\&D_1 = \frac{-D + \sqrt{D^2 + \dfrac{8L}{RT}}}{2}\end{aligned} \tag{8.84}$$

So, the output voltage is

$$V_o = V_s\left(\frac{D}{D+D_1}\right) = V_s\left(\frac{2D}{D+\sqrt{D^2+\frac{8L}{RT}}}\right) \tag{8.85}$$

For instance, for $V_s = 24\ V$, $L = 200\ \mu\text{H}$, $R = 20\ \Omega$, $C = 1000\ \mu\text{F}$, $f = 10\ \text{kHz}$, and $D = 0.4$, the output voltage is 13.9 V and $D_1 = 0.29$.

8.16 Other types of DC-DC converters operating in DCM

Other types of DC-DC converters can be analyzed in the same way like the buck converter. We summarize the important relations for some of the important DC-DC converters in this section.

8.16.1 Boost converter

For the boost converter shown in Fig. 8.39,

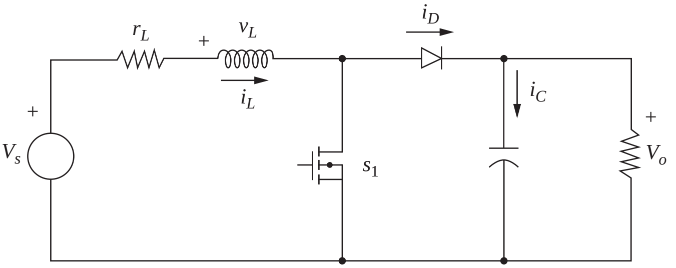


FIG. 8.39 Schematic of boost converter. r_L shows the inductor ESR.

The output voltage for CCM is

$$V_o = \left(\frac{V_s}{1-D}\right)\left(\frac{1}{1+\frac{r_L}{R(1-D)^2}}\right) \tag{8.86}$$

The output voltage for DCM is

$$V_o = \frac{1}{2}\left(1+\sqrt{1+\frac{2D^2RT}{L}}\right)V_s \tag{8.87}$$

L_{min} is (L_{min} is the minimum inductor value for CCM operation, i.e., converter is operated in CCM if $L > L_{min}$)

$$L_{min} = \frac{D(1-D)^2 R}{2f} \tag{8.88}$$

$\frac{\Delta V_o}{V_o}$ is

$$\frac{\Delta V_o}{V_o} = \frac{D}{RCf} \tag{8.89}$$

8.16.2 Buck-boost converter

For the buck-boost converter shown in Fig. 8.40,

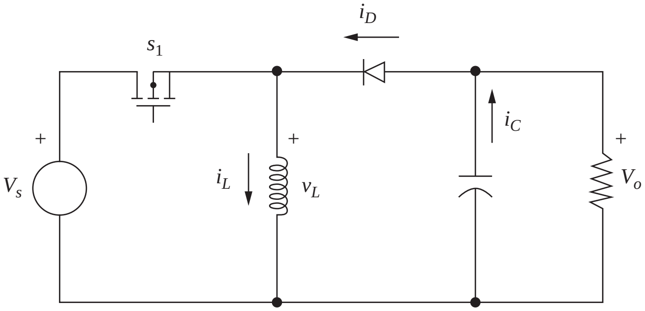


FIG. 8.40 Schematic of buck-boost converter.

The output voltage for CCM is (note to polarity reversal property of buck-boost converter)

$$V_o = -V_s\left(\frac{D}{1-D}\right) \tag{8.90}$$

L_{min} is

$$L_{min} = \frac{(1-D)^2 R}{2f} \tag{8.91}$$

$\frac{\Delta V_o}{V_o}$ is

$$\frac{\Delta V_o}{V_o} = \frac{D}{RCf} \tag{8.92}$$

8.16.3 Cuk converter

For the Cuk converter shown in Fig. 8.41,

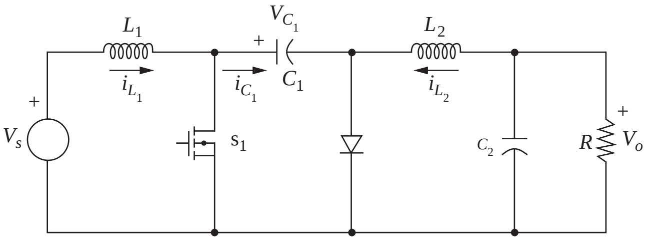

L_1
V_{C_1}
L_2
+
i_{L_1}
i_{C_1}
C_1
i_{L_2}
+
V_s
S_1
C_2
R
+
V_o

FIG. 8.41 Schematic of Cuk converter.

The output voltage for CCM is (note to polarity reversal property of Cuk converter)

$$V_o = -V_s\left(\frac{D}{1-D}\right) \tag{8.93}$$

$L_{1,min}$ is

$$L_{1,min} = \frac{(1-D)^2R}{2Df} \tag{8.94}$$

$L_{2,min}$ is

$$L_{2,min} = \frac{(1-D)R}{2f} \tag{8.95}$$

$\frac{\Delta V_o}{V_o}$ is

$$\frac{\Delta V_o}{V_o} = \frac{1-D}{8L_2C_2f^2} \tag{8.96}$$

8.16.4 Flyback converter

For the flyback converter shown in Fig. 8.42,

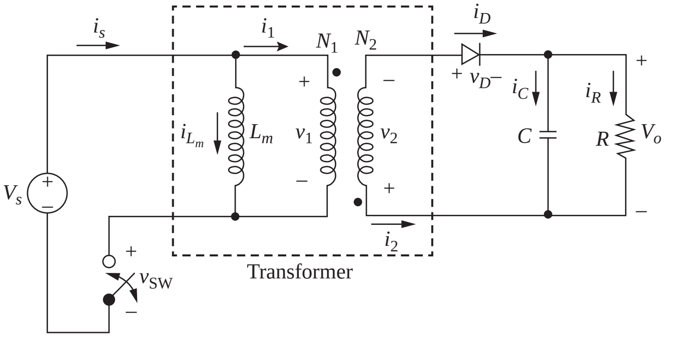

i_s
i_1
N_1
N_2
i_D
+
v_D
−
i_C
i_R
+
i_{L_m}
L_m
+
v_1
−
−
v_2
+
C
R
V_o
−
V_s
i_2
+
v_{SW}
−
Transformer

FIG. 8.42 Schematic of flyback converter.

The output voltage for CCM is

$$V_o = V_s\left(\frac{D}{1-D}\right)\left(\frac{N_2}{N_1}\right) \tag{8.97}$$

The output voltage for DCM is

$$V_o = V_s D\sqrt{\frac{R}{2L_m f}} \tag{8.98}$$

$L_{m,min}$ is ($L_{m,min}$ is the minimum magnetizing inductance required for CCM operation)

$$L_{m,min} = \frac{(1-D)^2 R}{2f}\left(\frac{N_1}{N_2}\right)^2 \tag{8.99}$$

$\frac{\Delta V_o}{V_o}$ is

$$\frac{\Delta V_o}{V_o} = \frac{D}{RCf} \tag{8.100}$$

8.17 Dynamics of DC-DC converters

In this section, we will extract the small signal dynamical model of DC-DC converters. This section studies the converters operating in CCM. Generally the converters operate in CCM; however, DCM has its own applications. Refer to [8] in order to extract the dynamical model of converters operate in DCM.

DC-DC converters are variable structure *nonlinear* dynamical systems. So, an LTI model is only an approximation. Although the obtained model is an approximation, it is well enough to start the controller design process.

In this section, we will introduce the State Space Averaging (SSA), which is one of the most important tools to model DC-DC converters. We will show how MATLAB® can do the mathematic machinery of SSA as well.

8.17.1 Overview of state space averaging (SSA)

Assume we want to compare two students. Students' marks are shown in Table 8.2.

TABLE 8.2 Hypothetical students' marks.

Student	Math (4 credits)	Physics (3 credits)	Biology (2 credits)
A	75	70	55
B	80	65	60

In order to compare them fairly, we must consider all the marks. This is done with the aid of averaging:

$$\begin{aligned} \mathrm{avg}_A &= \frac{75\times 4+70\times 3+55\times 2}{4+3+2} = 68.89 \\ \mathrm{avg}_B &= \frac{80\times 4+65\times 3+60\times 2}{4+3+2} = 70.55 \end{aligned} \tag{8.101}$$

So, second student is more successful since she or he has a higher average. Note that each mark is multiplied by the credits, so importance of the courses is entered into the averaging process.

The logic behind SSA is similar to the logic behind averaging marks. In this case, we average circuits instead of marks. Studying an example is quite helpful. Details and mathematic machinery will be shown in the next sections.

Assume a simple Buck converter like that shown in Fig. 8.43.

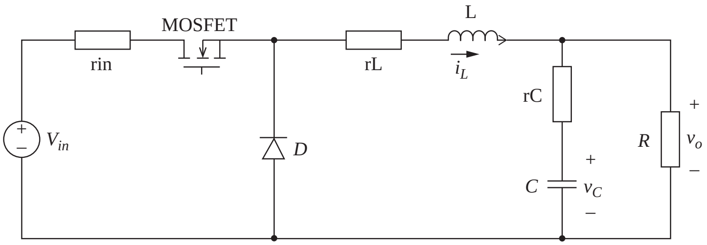


FIG. 8.43 Schematic of buck converter in presence of components ESRs.

Based on the MOSFET status (on or off), two equivalent subcircuits can be extracted as shown in Fig. 8.44.

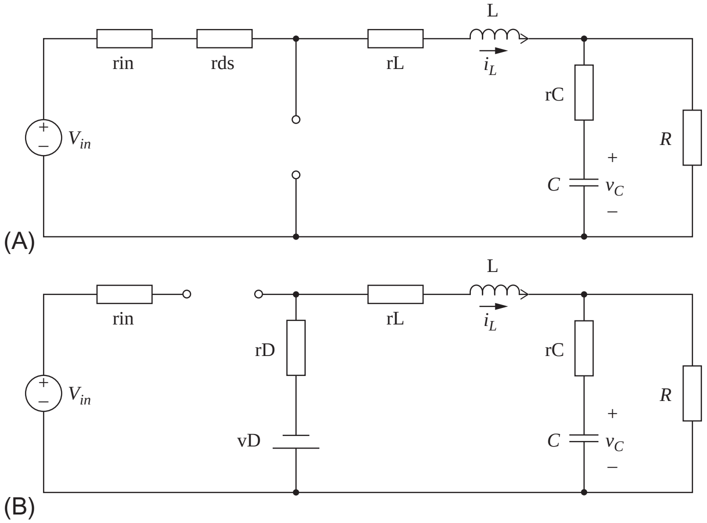


FIG. 8.44 Equivalent circuit for (A) closed MOSFET, (B) opened MOSFET.

We want to find a model for the Buck converter shown in Fig. 8.3.1, but we have two subcircuits. Each subcircuit has its own dynamic equation. We must find a way to average these two sets of equations.

Assume that the Buck converter shown in Fig. 8.43 spends 80% of the switching period in the MOSFET on state (Fig. 8.44A) and only 20% of switching period in the MOSFET off state (Fig 8.44B). In this case, the Buck converter spends most of its time in the MOSFET on state. So, it is logical to give a higher weight to MOSFET on equation set when averaging is done. It is not logical to average the equations (dynamical equations for MOSFET on and MOSFET off states) using the same weight.

SSA uses the percentage of switching time as the weights (credits in mark*averaging problem). For instance if the Buck converter spend 80% of switching time in the MOSFET on state and 20% of switching time in the MOSFET off state, then the MOSFET on equation set is multiplied by 0.8 and MOSFET off equation set is multiplied by 0.2.

We need an LTI model of the converter. So, the obtained average model must be linearized around the steady-state operating point. Differentiation is used to extract the linear model of converter. So, we can summarize the steps of SSA as below:

- Dynamical equations of all the subcircuits are extracted.
- Equations are averaged using the duty ratio as weight.
- Averaged equations are linearized around the operating point using differentiation.

We use the capital letters for steady-state values and tilde for small-signal perturbations. Small-signal perturbation is much smaller than the steady-state part. For example, the duty ratio of MOSFET gate signal is shown as $d = D + \tilde{d}$. This shows the duty ratio of MOSFET gate signal(d) is composed of two parts: steady-state part (D) and small-signal part $(\tilde{d})$. Small-signal component, as the name suggests, is quite smaller than the steady-state part, i.e., $\tilde{d} \ll D$.

8.17.2 Dynamical model of buck converter

Assume a buck converter like that shown in Fig. 8.45. rin, rL, and rC show the input source internal resistance, the inductor series resistance, and the capacitor series resistance, respectively. We assume values of elements are selected such that converter operates in CCM.

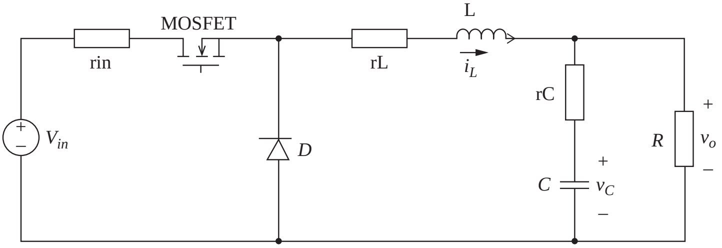


FIG. 8.45 Buck converter circuit. rin, rL, and rC show the internal resistance of the input DC source, inductor ESR, and capacitor ESR, respectively.

MOSFET is closed and opened with the aid of pulses shown in Fig. 8.46. When gate pulse is high, MOSFET is closed. According to Fig. 8.46, MOSFET is closed for duration of $d \times T$ seconds and is opened for duration of $T - d \times T = (1 - d) \times T$ seconds. T and d show switching period and duty ratio, respectively.

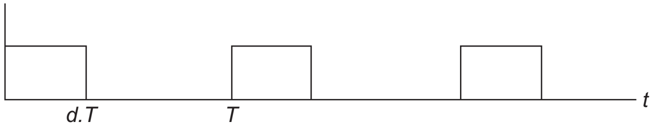


FIG. 8.46 MOSFET gate pulses.

Figs. 8.47 and 8.48 show the equivalent circuit for closed and opened MOSFET, respectively.

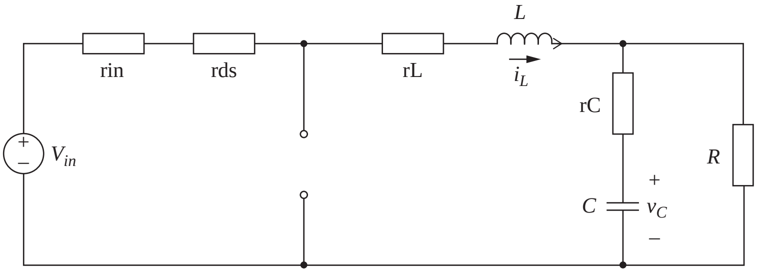


FIG. 8.47 Equivalent circuit for closed MOSFET. rds shows the resistance between drain and source of a closed MOSFET.

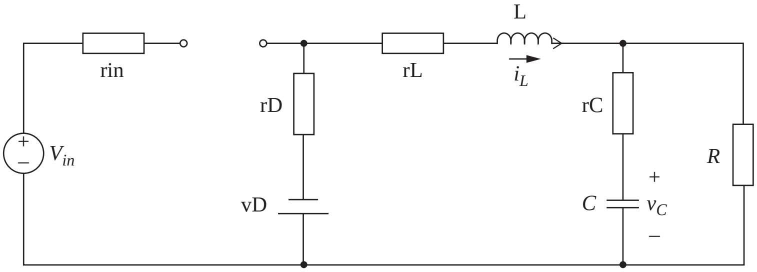


FIG. 8.48 Equivalent circuit for opened MOSFET. rD and vD show the diode series resistance and diode forward voltage drop, respectively.

These two linear circuits are used to obtain the converters transfer functions. First of all each circuit is analyzed separately.

8.17.2.1 Dynamical equations for closed MOSFET case

When MOSFET is closed, circuit equations can be written as:

$$\begin{cases} (r_{in}+r_{ds}+r_L)i_L+L\dfrac{di_L}{dt}+R\left(i_L-C\dfrac{dv_C}{dt}\right)=v_{in} \\ r_C C\dfrac{dv_C}{dt}+v_C=R\left(i_L-C\dfrac{dv_C}{dt}\right) \end{cases} \tag{8.102}$$

Equations can be simplified using simple algebraic manipulations,

$$\begin{cases} (r_{in}+r_{ds}+r_L+R)i_L+L\dfrac{di_L}{dt}-RC\dfrac{dv_C}{dt}=v_{in} \\ (r_C+R)C\dfrac{dv_C}{dt}+v_C=Ri_L \end{cases} \tag{8.103}$$

$$\begin{cases} (r_{in}+r_{ds}+r_L+R)i_L+L\dfrac{di_L}{dt}-\dfrac{R}{R+r_C}(Ri_L-v_C)=v_{in} \\ C\dfrac{dv_C}{dt}=\dfrac{1}{R+r_C}(Ri_L-v_C) \end{cases} \tag{8.104}$$

$$\begin{cases} L\dfrac{di_L}{dt}=-(r_{in}+r_{ds}+r_L+R)i_L+\dfrac{R^2}{R+r_C}i_L-\dfrac{R}{R+r_C}v_C+v_{in} \\ C\dfrac{dv_C}{dt}=\dfrac{1}{R+r_C}(Ri_L-v_C) \end{cases} \tag{8.105}$$

Equation of output load voltage can be written as,

$$v_o=R\left(i_L-C\frac{dv_C}{dt}\right)=R\left(\frac{r_C}{r_C+R}i_L+\frac{1}{R+r_C}v_C\right) \tag{8.106}$$

8.17.2.2 Dynamical equations for opened MOSFET case

When MOSFET is opened, circuit equation can be written as:

$$\begin{cases} V_D+(r_D+r_L)i_L+L\dfrac{di_L(t)}{dt}+r_C C\dfrac{dv_C}{dt}+v_C=0 \\ r_C C\dfrac{dv_C}{dt}+v_C=R\left(i_L-C\dfrac{dv_C}{dt}\right) \end{cases} \tag{8.107}$$

Equations can be simplified using simple algebraic manipulations,

$$\begin{cases} V_D+(r_D+r_L)i_L+L\dfrac{di_L(t)}{dt}+\dfrac{R.r_C}{R+r_C}i_L+\dfrac{R}{R+r_C}v_C=0 \\ C\dfrac{dv_C}{dt}=\dfrac{R}{R+r_C}i_L-\dfrac{1}{R+r_C}v_C \end{cases} \tag{8.108}$$

$$\begin{cases} L\dfrac{di_L(t)}{dt} = -\left(r_D + r_L + \dfrac{R.r_C}{R+r_C}\right)i_L - \dfrac{R}{R+r_C}v_C - V_D \\ C\dfrac{dv_C}{dt} = \dfrac{R}{R+r_C}i_L - \dfrac{1}{R+r_C}v_C \end{cases} \tag{8.109}$$

Equation of output load voltage can be written as,

$$v_o = R\left(i_L - C\frac{dv_C}{dt}\right) = R\left(\frac{r_C}{r_C+R}i_L + \frac{1}{R+r_C}v_C\right) \tag{8.110}$$

8.17.2.3 Averaging

Results of previous analysis are superimposed using averaging. MOSFET is closed for $d.\ T$ seconds and is opened for $(1-d).\ T$ seconds. So, it is fair to multiply the equations obtained for closed MOSFET by $d.\ T$ and the equations obtained for opened MOSFET by $(1-d).\ T$ and average over one periode (duration of one period is T seconds).

We obtain the inductor current equations as:

$$\begin{cases} L\dfrac{di_L}{dt} = -\left(r_{in} + r_{ds} + r_L + R - \dfrac{R^2}{R+r_C}\right)i_L - \dfrac{R}{R+r_C}v_C + v_{in}, \text{For closed MOSFET} \\ L\dfrac{di_L(t)}{dt} = -\left(r_D + r_L + \dfrac{R.r_C}{R+r_C}\right)i_L - \dfrac{R}{R+r_C}v_C - V_D, \qquad \text{For opened MOSFET} \end{cases} \tag{8.111}$$

Left and right sides of closed MOSFET and opened MOSFET equations are multiplied by $d.\ T$ and $(1-d).\ T$, respectively.

$$\begin{cases} L\dfrac{di_L}{dt} \times d.T = \left(-(r_{in} + r_{ds} + r_L + R)i_L + \dfrac{R^2}{R+r_C}i_L - \dfrac{R}{R+r_C}v_C + v_{in}\right) \times d.T \\ L\dfrac{di_L(t)}{dt} \times (1-d).T = \left(-\left(r_D + r_L + \dfrac{R.r_C}{R+r_C}\right)i_L - \dfrac{R}{R+r_C}v_C - V_D\right) \times (1-d).T \end{cases} \tag{8.112}$$

Corresponding sides are added together:

$$\begin{aligned} &d.T \times L\frac{di_L}{dt} + (1-d).T \times L\frac{di_L}{dt} \\ &\quad = d.T \times \left(-(r_{in} + r_{ds} + r_L + R)i_L + \frac{R^2}{R+r_C}i_L - \frac{R}{R+r_C}v_C + v_{in}\right) + (1-d).T \\ &\quad\quad \times \left(-\left(r_D + r_L + \frac{R.r_C}{R+r_C}\right)i_L - \frac{R}{R+r_C}v_C - V_D\right) \end{aligned} \tag{8.113}$$

Averaging is realized by multiplying both sides by $\frac{1}{T}$:

$$\begin{aligned} d\times L\frac{di_L}{dt}+(1-d)\times L\frac{di_L}{dt} \\ &= d\times\left(-(r_{in}+r_{ds}+r_L+R)i_L+\frac{R^2}{R+r_C}i_L-\frac{R}{R+r_C}v_C+v_{in}\right) \\ &+(1-d)\times\left(-\left(r_D+r_L+\frac{R.r_C}{R+r_C}\right)i_L-\frac{R}{R+r_C}v_C-V_D\right) \end{aligned} \tag{8.114}$$

One can multiply both sides of closed MOSFET and opened MOSFET equations by d and $1-d$, respectively, and add the corresponding sides together. The result is the same as the one obtained above. In the next analysis, we will multiply the equations by d and $1-d$.

After some simple algebraic manipulations,

$$\begin{aligned} L\frac{di_L}{dt} &= -d\times\left(r_{in}+r_{ds}+r_L+R-\frac{R^2}{R+r_C}\right)i_L \\ &-(1-d)\times\left(r_D+r_L+\frac{R.r_C}{R+r_C}\right)i_L-\frac{R}{R+r_C}v_C-(1-d)V_D+dv_{in} \end{aligned}$$

$$L\frac{di_L}{dt}=-d\times R_1 i_L-(1-d)\times R_2 i_L-\frac{R}{R+r_C}v_C-(1-d)V_D+dv_{in} \tag{8.115}$$

where

$$R_1=r_{in}+r_{ds}+r_L+R-\frac{R^2}{R+r_C} \tag{8.116}$$

and

$$R_2=r_D+r_L+\frac{R.r_C}{R+r_C} \tag{8.117}$$

The same procedure can be applied to the capacitor voltage equations. We obtain the capacitor voltage equations as:

$$\begin{cases} C\dfrac{dv_C}{dt}=\dfrac{R}{R+r_C}i_L-\dfrac{1}{R+r_C}v_C & \text{For closed MOSFET} \\ C\dfrac{dv_C}{dt}=\dfrac{R}{R+r_C}i_L-\dfrac{1}{R+r_C}v_C & \text{For opened MOSFET} \end{cases} \tag{8.118}$$

Left and right sides of closed MOSFET and opened MOSFET equations are multiplied by d and $(1-d)$, respectively, and the corresponding sides are added to each other.

$$d\times C\frac{dv_C}{dt}+(1-d)\times C\frac{dv_C}{dt}=d\times\left(\frac{R}{R+r_C}i_L-\frac{1}{R+r_C}v_C\right)+(1-d)\left(\frac{R}{R+r_C}i_L-\frac{1}{R+r_C}v_C\right)$$

$$C\frac{dv_C}{dt}=\left(\frac{R}{R+r_C}i_L-\frac{1}{R+r_C}v_C\right) \tag{8.119}$$

So the equation of *average* system can be written as:

$$\begin{cases} L\dfrac{di_L}{dt} = -d\times R_1 i_L - (1-d)\times R_2 i_L - \dfrac{R}{R+r_C}v_C - (1-d)V_D + dv_{in} \\ C\dfrac{dv_C}{dt} = \dfrac{R}{R+r_C}i_L - \dfrac{1}{R+r_C}v_C \end{cases} \tag{8.120}$$

where

$$R_1 = r_{in} + r_{ds} + r_L + R - \frac{R^2}{R+r_C}$$

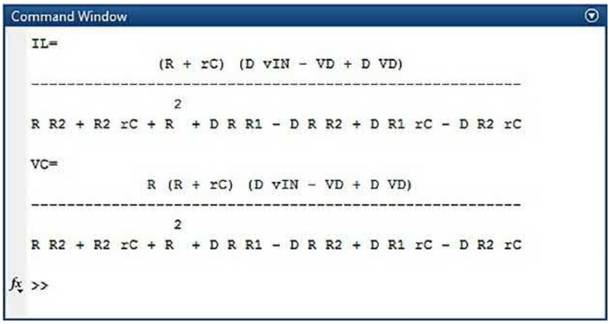
$$R_2 = r_D + r_L + \frac{R.r_C}{R+r_C}$$

This average system can be used to obtain the steady-state currents and voltages. To obtain the steady-state currents and voltages, one must replace the left-hand side with zero (remember that once the system reaches its steady state, the changes, i.e., derivative of states, are zero). Capital letters show the steady-state values. For instance, I_L shows the steady-state inductor current.

$$\begin{cases} 0 = -D\times R_1 I_L - (1-D)\times R_2 I_L + \dfrac{R}{R+r_C}V_C - (1-D)V_D + DV_{IN} \\ 0 = \dfrac{R}{R+r_C}I_L - \dfrac{1}{R+r_C}V_C \end{cases} \tag{8.121}$$

MATLAB can be used to solve the obtained equation set. The following code solves the obtained equation set. Solution is shown in Fig. 8.49.

```
clc
clear all

syms R1 R2 R D IL VC rC rL VD vIN

eq1=-D*R1*IL-(1-D)*R2*IL-R/(R+rC)*VC-(1-D)*VD+D*vIN;
eq2=R/(R+rC)*IL-1/(R+rC)*VC;

DC_operatingPoint=solve(eq1,eq2,'[IL VC]');

disp('IL=')
pretty(simplify(DC_operatingPoint.IL))

disp('VC=')
pretty(simplify(DC_operatingPoint.VC))
```



```
IL=
                (R + rC) (D vIN - VD + D VD)
---------------------------------------------------------
                 2
R R2 + R2 rC + R  + D R R1 - D R R2 + D R1 rC - D R2 rC

VC=
              R (R + rC) (D vIN - VD + D VD)
---------------------------------------------------------
                 2
R R2 + R2 rC + R  + D R R1 - D R R2 + D R1 rC - D R2 rC

fx >>
```

FIG. 8.49 The steady-state current and voltage.

So the steady-state values of inductor current and capacitor voltage are:

$$\begin{cases} I_L = \dfrac{((R+r_C)(DV_{IN}-(1-D)V_D}{(R+r_C)R_2+R^2+D(R+r_C)(R_1-R_2)} \\ V_C = \dfrac{((R+r_C)(DV_{IN}-(1-D)V_D}{(R+r_C)R_2+(1-2D)R^2+D(R+r_C)(R_1-R_2)}\times R \end{cases} \tag{8.122}$$

If we ignore rin, rds, rD, VD (i.e., rin= rds= rD= VD= 0) steady-state values are obtained as:

$$\begin{cases} I_L = \dfrac{D\times V_{IN}}{R} \\ V_C = D\times V_{IN} \end{cases} \tag{8.123}$$

which is the familiar equations for ideal (i.e., a converter with 100% efficiency) buck converter operating in CCM.

Averaging procedure must be applied to the output equation as well.

$$\begin{aligned} d\times v_o+(1-d)\times v_o &= d\times R\left(\frac{r_C}{r_C+R}i_L+\frac{1}{R+r_C}v_C\right)+(1-d) \\ &\quad \times R\left(\frac{r_C}{r_C+R}i_L+\frac{1}{R+r_C}v_C\right) \end{aligned} \tag{8.124}$$

After some simple algebraic manipulations,

$$v_o = R\left(\frac{r_C}{r_C+R}i_L+\frac{1}{R+r_C}v_C\right) \tag{8.125}$$

is obtained.

8.17.2.4 Linearization of averaged equations

Averaged equations are obtained as:

$$\begin{cases} L\dfrac{di_L}{dt} = -d\times R_1 i_L-(1-d)\times R_2 i_L-\dfrac{R}{R+r_C}v_C-(1-d)V_D+dv_{in} \\ C\dfrac{dv_C}{dt} = \dfrac{R}{R+r_C}i_L-\dfrac{1}{R+r_C}v_C \end{cases} \tag{8.126}$$

where

$$R_1 = r_{in} + r_{ds} + r_L + R - \frac{R^2}{R + r_C}$$

$$R_2 = r_D + r_L + \frac{R \times r_C}{R + r_C}$$

Linearization is done using the Taylor series. Assume that,

$$\begin{aligned} i_L &= I_L + \widetilde{i}_L \\ v_C &= V_C + \widetilde{v}_C \\ d &= D + \widetilde{d} \end{aligned} \tag{8.127}$$

where $\widetilde{i}_L \ll I_L$, $\widetilde{v}_C \ll V_C$ and $\widetilde{d} \ll D$. Diode forward voltage drop has been assumed to be constant. These equations show that each variable has a steady steady-state value (I_L, V_C, and D) and a small perturbation ($\widetilde{i}_L, \widetilde{v}_C$, and $\widetilde{d}$). Steady-state values are obtained by solving the averaged equations with left-hand side equal to zero (see Fig. 8.49). Steady-state values are much bigger than the small-signal perturbation part.

We substitute these new variables into the equations,

$$\begin{aligned} L\frac{d\left(I_L + \widetilde{i}_L\right)}{dt} = &-\left(D + \widetilde{d}\right) \times R_1\left(I_L + \widetilde{i}_L\right) - \left(1 - \left(D + \widetilde{d}\right)\right) \times R_2\left(I_L + \widetilde{i}_L\right) \\ &- \frac{R}{R + r_C}\left(V_C + \widetilde{v}_C\right) - \left(1 - \left(D + \widetilde{d}\right)\right)V_D + \left(D + \widetilde{d}\right)\left(V_{IN} + \widetilde{v}_{in}\right) \end{aligned} \tag{8.128}$$

$$C\frac{d\left(V_C + \widetilde{v}_C\right)}{dt} = \frac{R}{R + r_C}\left(I_L + \widetilde{i}_L\right) - \frac{1}{R + r_C}\left(V_C + \widetilde{v}_C\right) \tag{8.129}$$

After simple algebraic manipulations,

$$\begin{aligned} L\frac{d\left(I_L + \widetilde{i}_L\right)}{dt} = &-\left(D + \widetilde{d}\right) \times R_1\left(I_L + \widetilde{i}_L\right) - \left(1 - \left(D + \widetilde{d}\right)\right) \times R_2\left(I_L + \widetilde{i}_L\right) \\ &- \frac{R}{R + r_C}\left(V_C + \widetilde{v}_C\right) - \left(1 - \left(D + \widetilde{d}\right)\right)V_D + \left(D + \widetilde{d}\right)\left(V_{IN} + \widetilde{v}_{in}\right) \Rightarrow \\ L\frac{d\left(I_L + \widetilde{i}_L\right)}{dt} = &-R_1DI_L - R_1D\widetilde{i}_L - R_1I_L\widetilde{d} - R_1\widetilde{i}_L\widetilde{d} + R_2(D-1)I_L + R_2(D-1)\widetilde{i}_L + R_2I_L\widetilde{d} + R_2\widetilde{i}_L\widetilde{d} \\ &- \frac{R}{R + r_C}V_C - \frac{R}{R + r_C}\widetilde{v}_c + (D-1)V_D + V_D\widetilde{d} + DV_{IN} + D\widetilde{v}_{in} + V_{IN}\widetilde{d} + \widetilde{v}_{in}\widetilde{d} \Rightarrow \\ L\frac{d\left(I_L + \widetilde{i}_L\right)}{dt} = &-R_1DI_L + R_2(D-1)I_L + (D-1)V_D - \frac{R}{R + r_C}V_C + DV_{IN} + \widetilde{v}_{in}\widetilde{d} + R_2\widetilde{i}_L\widetilde{d} \\ &- R_1\widetilde{i}_L\widetilde{d} + (R_2(D-1) - R_1D)\widetilde{i}_L - \frac{R}{R + r_C}\widetilde{v}_c + (V_{IN} + V_D + (R_2 - R_1)I_L)\widetilde{d} + D\widetilde{v}_{in} \end{aligned} \tag{8.130}$$

is obtained. $L\frac{d\left(I_L + \widetilde{i}_L\right)}{dt} = L\frac{d\left(\widetilde{i}_L\right)}{dt}$ since derivative of a constant term (I_L) is zero. Right-hand side can be grouped into three:

- $-R_1DI_L+R_2(D-1)I_L+(D-1)V_D-\frac{R}{R+r_C}V_C+DV_{IN}$
- $\tilde{v}_{in}\tilde{d}+R_2\tilde{i}_L\tilde{d}-R_1\tilde{i}_L\tilde{d}$
- $(R_2(D-1)-R_1D)\tilde{i}_L-\frac{R}{R+r_C}\tilde{v}_c+(V_{IN}+V_D+(R_2-R_1)I_L)\tilde{d}+D\tilde{v}_{in}$

If we substitute the steady-state values obtained before (Fig. 8.49) into the $-R_1DI_L+R_2(D-1)I_L+(D-1)V_D-\frac{R}{R+r_C}V_C+DV_{IN}$, the result will become 0. Second group can vanish as well because multiplication of two small numbers is a small number, as well. So, only terms of third group are important.

$$L\frac{d\left(\tilde{i}_L\right)}{dt}\approx(R_2(D-1)-R_1D)\tilde{i}_L-\frac{R}{R+r_C}\tilde{v}_c+(V_{IN}+V_D+(R_2-R_1)I_L)\tilde{d}+D\tilde{v}_{in} \tag{8.131}$$

The same procedure can be applied to the averaged capacitor voltage equation:

$$\begin{aligned}C\frac{d(V_C+\tilde{v}_C)}{dt}&=\frac{R}{R+r_C}\left(I_L+\tilde{i}_L\right)-\frac{1}{R+r_C}(V_C+\tilde{v}_C)\Longrightarrow C\frac{d(V_C+\tilde{v}_C)}{dt}\\&=\frac{R}{R+r_C}I_L-\frac{1}{R+r_C}V_C+\frac{R}{R+r_C}\tilde{i}_L-\frac{1}{R+r_C}\tilde{v}_C\end{aligned} \tag{8.132}$$

V_C is constant, so its derivative is zero. So,

$$C\frac{d(\tilde{v}_C)}{dt}=\frac{R}{R+r_C}I_L-\frac{1}{R+r_C}V_C+\frac{R}{R+r_C}\tilde{i}_L-\frac{1}{R+r_C}\tilde{v}_C \tag{8.133}$$

Right-hand side can be grouped into two groups:

- $\frac{R}{R+r_C}I_L-\frac{1}{R+r_C}V_C$
- $\frac{R}{R+r_C}\tilde{i}_L-\frac{1}{R+r_C}\tilde{v}_C$

If we substitute the steady-state values obtained for I_L and V_C into the $\frac{R}{R+r_C}I_L-\frac{1}{R+r_C}V_C$, the result will be 0. So, only terms of second group are important.

$$C\frac{d(\tilde{v}_C)}{dt}\approx\frac{R}{R+r_C}\tilde{i}_L-\frac{1}{R+r_C}\tilde{v}_C \tag{8.134}$$

Output equation is linearized in the same way. Output voltage (v_o) is decomposed into two components: $v_o=V_o+\tilde{v}_o$. V_o shows the large-signal component and $\tilde{v}_o$ shows the small-signal component

$$\begin{aligned}v_o&=R\left(\frac{r_C}{r_C+R}i_L+\frac{1}{R+r_C}v_C\right)\Rightarrow\\V_o+\tilde{v}_o&=R\left(\frac{r_C}{r_C+R}\left(I_L+\tilde{i}_L\right)+\frac{1}{R+r_C}(V_C+\tilde{v}_C)\right)\Rightarrow\\V_o+\tilde{v}_o&=\frac{R.r_C}{r_C+R}I_L+\frac{R}{R+r_C}V_C+\frac{R.r_C}{r_C+R}\tilde{i}_L+\frac{R}{R+r_C}\tilde{v}_C\end{aligned} \tag{8.135}$$

Steady-state value of load voltage can be obtained as:

$$V_o = \frac{R.r_C}{r_C + R} I_L + \frac{R}{R + r_C} V_C \tag{8.136}$$

The small-signal variation of output voltage is:

$$\tilde{v}_o = \frac{R.r_C}{r_C + R} \tilde{i}_L + \frac{R}{R + r_C} \tilde{v}_C \tag{8.137}$$

So, linearized small-signal model of buck converter can be written as:

$$\begin{cases} \dfrac{d\left(\tilde{i}_L\right)}{dt} \approx \dfrac{1}{L}\left[(R_2(D-1) - R_1 D)\tilde{i}_L - \dfrac{R}{R + r_C}\tilde{v}_c + (V_{IN} + V_D + (R_2 - R_1)I_L)\,\tilde{d} + D\tilde{v}_{in}\right] \\ \dfrac{d\left(\tilde{v}_C\right)}{dt} \approx \dfrac{1}{C}\left[\dfrac{R}{R + r_C}\tilde{i}_L - \dfrac{1}{R + r_C}\tilde{v}_C\right] \end{cases}$$

$$\tilde{v}_o = \frac{R.r_C}{r_C + R} \tilde{i}_L + \frac{R}{R + r_C} \tilde{v}_C \tag{8.138}$$

where

$$R_1 = r_{in} + r_{ds} + r_L + R - \frac{R^2}{R + r_C}$$

$$R_2 = r_D + r_L + \frac{R \times r_C}{R + r_C}$$

It can be written in the form of a state space equation (we used notation $\mathbb{C}x$ instead of Cx, since C is used for capacitor. $\mathbb{C}$ is a matrix.):

$$\begin{cases} \dot{x} = Ax + Bu \\ y = \mathbb{C}x \end{cases} \tag{8.139}$$

where

$$x = \begin{bmatrix} \tilde{i}_L \\ \tilde{v}_c \end{bmatrix}$$

$$u = \begin{bmatrix} \tilde{d} \\ \tilde{v}_{in} \end{bmatrix}$$

$$y = v_o$$

$$A = \begin{bmatrix} \dfrac{R_2(D-1) - R_1 D}{L} & -\dfrac{R}{(R + r_C)L} \\ \dfrac{R}{(R + r_C)C} & -\dfrac{1}{(R + r_C)C} \end{bmatrix} \tag{8.140}$$

$$B = \begin{bmatrix} \dfrac{(V_{IN} + V_D + (R_2 - R_1)I_L)}{L} & \dfrac{D}{L} \\ \dfrac{0}{C} & \dfrac{0}{C} \end{bmatrix}$$

$$C = \begin{bmatrix} \dfrac{R.r_C}{r_C + R} & \dfrac{R}{r_C + R} \end{bmatrix}$$

Assume a buck converter with the following parameters:

R= 5 Ω, Vin=50 V, rin=0.1 Ω, L=400 μH, rL=0.1 Ω, C=100 μF, rC=0.05 Ω, D=0.41, rds=0.1 Ω, rD=0.1 Ω, and VD=0.7 V.

Following program calculates the small-signal transfer functions according to the given values. After running the program, $\frac{\tilde{v}_o(s)}{\tilde{d}(s)} = \frac{6184s + 1.237\times 10^9}{s^2 + 2574s + 2.568\times 10^7}$ and $\frac{\tilde{v}_o(s)}{\tilde{v}_{in}(s)} = \frac{50.74s + 1.015\times 10^7}{s^2 + 2574s + 2.568\times 10^7}$ are obtained. Bode plots of these transfer functions are shown in Figs. 8.50 and 8.51.

```
%This program calculate the small signal transfer
%functions for Buck converter
R=5;

VIN=50;
rin=.1;

L=400e-6;
rL=.1;

C=100e-6;
rC=.05;

rD=.01;
VD=.7;

rds=.1;

D=.41;

R1=rin+rds+rL+R*rC/(R+rC);
R2=rD+rL+R*rC/(R+rC);

IL=(R+rC)*(D*VIN-(1-D)*VD)/((R+rC)*R2+R^2+D*(R+rC)*(R1-R2));

A=[(R2*(D-1)-R1*D)/L -R/(R+rC)/L;R/(R+rC)/C -1/(R+rC)/C];
B=[(VIN+VD+(R2-R1)*IL)/L D/L;0 0];
CC=[R*rC/(rC+R) R/(R+rC)]; %C shows the capacitance so CC is used
for matrix
H=tf(ss(A,B,CC,0));
v0_d=H(1)% transfer function between output voltage and duty ratio
v0_vin=H(2) %transfer function between output voltage and input
source
figure(1)
bode(v0_d), grid on
figure(2)
bode(v0_vin), grid on
```

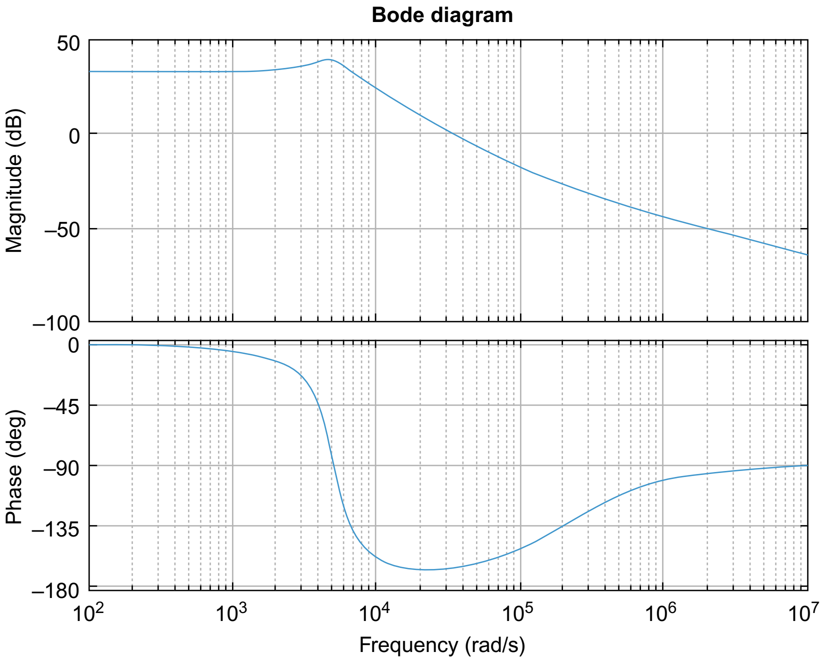


FIG. 8.50 Bode plot of $\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$ for studied buck converter.

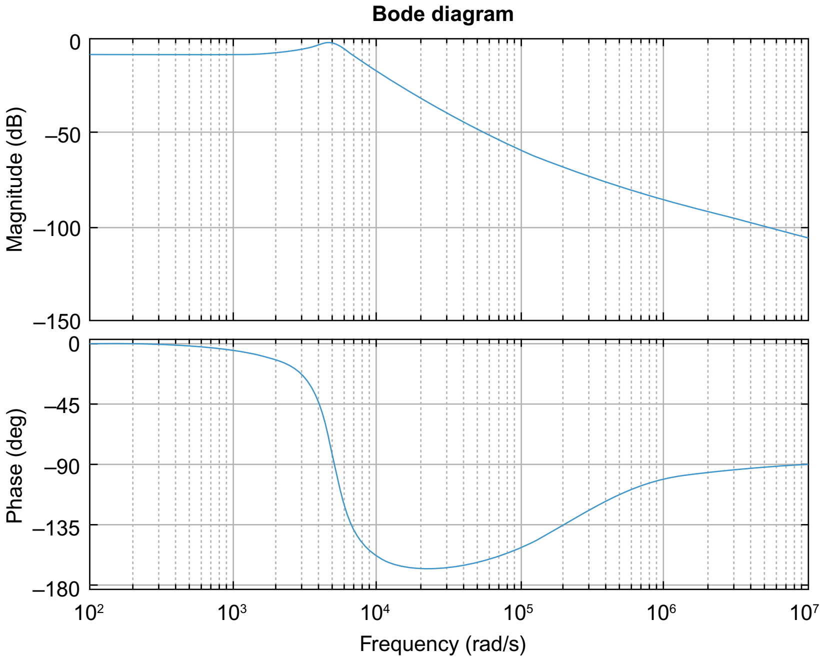


FIG. 8.51 Bode plot of audio susceptibility ($\frac{\tilde{v}_o(s)}{\tilde{v}_{in}(s)}$) for studied buck converter.

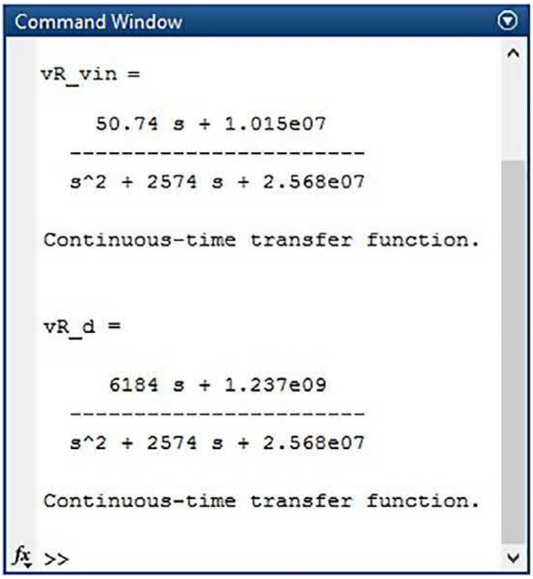
8.17.2.5 Obtaining the small-signal transfer functions using MATLAB®

MATLAB® can do the mathematical machinery of SSA easily without any error. Following program shows how MATLAB® can be used to extract the small-signal transfer functions of a buck converter. Outputs are shown in Fig. 8.52. As you see, results are the same with the previous analysis.

```
% This program extract the small signal transfer function
clc
clear all;
% Elements values
R=5;      %Load resistor
VIN=50;   %Input source voltage
rin=.1;   %Input source internal resistance
L=400e-6;%inductor
rL=.1;    %inductor series resistance
C=100e-6;%capacitor
rC=.05;   %capacitor series resistance
rD=.01;   %Diode series resistance
VD=.7;    %Diode forward voltage drop
rds=.1;   %MOSFET on resistance
D=.41;    %Duty ratio

% Symbolic variables
%iL: inductor current
%vC: capacitor voltage
%vin: input voltage source
%vD: diode forward voltage drop
%d: duty cycle
syms iL vC vin vD d

%CLOSED MOSFET EQUATIONS
M1=(-(rin+rds+rL+(R*rC/(R+rC)))*iL-R/(R+rC)*vC+vin)/L;%d(iL)/
dt for closed MOSFET
M2=(R/(R+rC)*iL-1/(R+rC)*vC)/C;           %d(vC)/dt for closed
MOSFET
vO1=R*(rC/(rC+R)*iL+1/(R+rC)*vC);
%OPENED MOSFET EQUATIONS
M3=(-(rD+rL+R*rC/(R+rC))*iL-R/(R+rC)*vC-vD)/L;  %d(iL)/dt for opened
MOSFET
M4=(R/(R+rC)*iL-1/(R+rC)*vC)/C;           %%d(vC)/dt  for  opened
MOSFET
vO2=R*(rC/(rC+R)*iL+1/(R+rC)*vC);
%AVERAGING
MA1= simplify(d*M1+(1-d)*M3);
```

```
MA2= simplify(d*M2+(1-d)*M4);
v0= simplify(d*v01+(1-d)*v02);
% DC OPERATİNG POİNT CALCULATİON
MA_DC_1=subs(MA1,[vin vD d],[VIN VD D]);
MA_DC_2=subs(MA2,[vin vD d],[VIN VD D]);

DC_SOL= solve(MA_DC_1==0,MA_DC_2==0,'iL','vC');

IL=eval(DC_SOL.iL); %IL is the inductor current steady state value
VC=eval(DC_SOL.vC); %VC is the capacitor current steady state value

%LINEARIZATION
% .
% x=Ax+Bu
%vector x=[iL;vC] is assumed. vector x is states.
%u=[vin;d] where vin=input voltage source and d=duty. vector u is
system inputs.
%
A11=subs(simplify(diff(MA1,iL)),[iL vC d vD],[IL VC D VD]);
A12=subs(simplify(diff(MA1,vC)),[iL vC d vD],[IL VC D VD]);

A21=subs(simplify(diff(MA2,iL)),[iL vC d vD],[IL VC D VD]);
A22=subs(simplify(diff(MA2,vC)),[iL vC d vD],[IL VC D VD]);

A=eval([A11 A12;
        A21 A22]); %variable A is matrix A in state space equation

B11=subs(simplify(diff(MA1,vin)),[iL vC d vD vin],[IL VC D VD VIN]);
B12=subs(simplify(diff(MA1,d)),[iL vC d vD vin],[IL VC D VD VIN]);

B21=subs(simplify(diff(MA2,vin)),[iL vC d vD vin],[IL VC D VD VIN]);
B22=subs(simplify(diff(MA2,d)),[iL vC d vD vin],[IL VC D VD VIN]);

B=eval([B11 B12;
        B21 B22]); % variable B is matrix B in state space equation

CC1=subs(simplify(diff(v0,iL)),[iL vC d vD],[IL VC D VD]);
CC2=subs(simplify(diff(v0,vC)),[iL vC d vD],[IL VC D VD]);
CC=eval([CC1 CC2]); %variable CC is matrix C in state space equation
                    % variable D shows duty so DD is used.
DD11=subs(simplify(diff(v0,vin)),[iL vC d vD vin],[IL VC D VD VIN]);
DD12=subs(simplify(diff(v0,d)),[iL vC d vD vin],[IL VC D VD VIN]);

DD=eval([DD11 DD12]); % variable DD is matrix D in state space equation
                      % variable D shows duty so DD is used.

H=tf(ss(A,B,CC,DD));

                %transfer function between input source and load
resistor voltage
```

```
                  % ~
vR_vin=H(1,1)     % vR(s)
                  % —
                  % ~
                  % vin(s)

                  %transfer function between duty ratio and load resistor
voltage
                  %~
vR_d=H(1,2)       %vR(s)
                  %-—
                  %~
                  %d(s)
```



```
vR_vin =

    50.74 s + 1.015e07
  ------------------------
  s^2 + 2574 s + 2.568e07

Continuous-time transfer function.


vR_d =

     6184 s + 1.237e09
  ------------------------
  s^2 + 2574 s + 2.568e07

Continuous-time transfer function.

>>
```

FIG. 8.52 Calculated transfer functions. `vR_vin` keeps the $\frac{\tilde{v}_o(s)}{\tilde{v}_{in}(s)}$ transfer function and `vR_d` keeps the $\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$ transfer function. $\tilde{v}_o(s)$ shows the small-signal changes of output load R.

The program is composed of three parts:

- Taking the parameter values,
- Applying averaging and linearization to the converters dynamical equations,
- Extracting transfer functions.

First few lines of code take the parameters values. This is quite useful and is recommended since you can run the code easily for different values of components.
]

Converters dynamical equations (variables `M1, M2, v01, M3, M4,` and `v02` in the code) must be extracted manually using KVL and KCL. MATLAB® can do symbolic computation using `syms` command. For more information on the command, type `help syms` in MATLAB's command line.

Variables MA1, MA2, and vO in the code are averaged variables. As stated earlier, the steady-state currents and voltages can be found by equating the derivative terms equal to zero. IL and VC are steady-state values of inductor current and capacitor voltage, respectively. These variables determine the steady-state operating point of converter.

The derivative is used to calculate the matrices of state space representation. State space representation has been converted into transfer function form using `tf` command.

The shown program can be used to extract boost or buck-boost dynamical equations as well. Only dynamical equations (variables `M1, M2, v01, M3, M4,` and `v02` in the code) must be rewritten according to the selected topology. Other parts of code remain unchanged.

8.18 PID controller design for converter

PID controllers are the most important type of controllers. In this section, we show how a PID controller can be designed for obtained control-to-output ($\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$) transfer function. The following small-signal transfer functions are obtained in the previous section:

$$\frac{\tilde{v}_o(s)}{\tilde{v}_{in}(s)} = \frac{50.74s + 1.015\times10^7}{s^2 + 2574s + 2.568\times10^7}$$

$$\frac{\tilde{v}_o(s)}{\tilde{d}(s)} = \frac{6184s + 1.237\times10^9}{s^2 + 2574s + 2.568\times10^7}$$

The following block diagram can be drawn for the studied buck converter (Fig. 8.53).

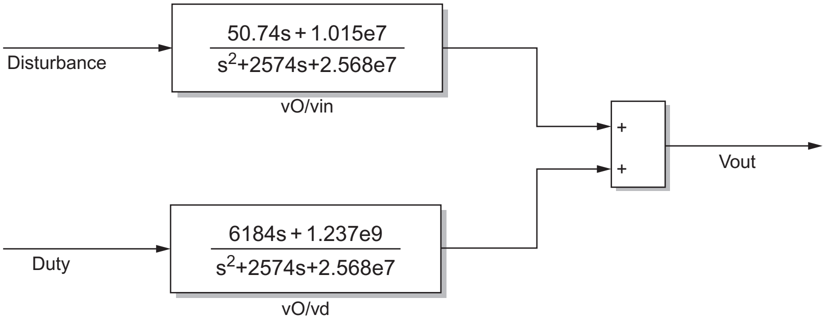


FIG. 8.53 Dynamical model of buck converter. The changes in duty ratio ($\tilde{d}$) and input voltage ($\tilde{v}_{in}$) affect the output voltage. The output current changes ($\tilde{i}_o$, see next section) will affect the output voltage, as well. However, the output current changes is ignored in the drawn block diagram.

The controller sets the duty cycle of converter, i.e., duty cycle is the control input. The input voltage changes $(\tilde{v}_{in}(s))$ play the role disturbance. Note that in the previous analysis we neglected the output load changes, i.e., we assumed the output load is constant. The change of output load current $(\tilde{i}_o)$ is another source of disturbance; however, it is neglected in this section.

Structure of control loop is shown in Fig. 8.54.

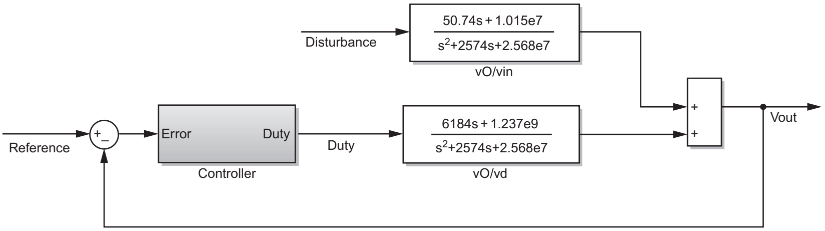
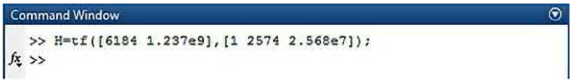


FIG. 8.54 Closed-loop control of buck converter. Controller block sets the duty ratio. The input voltage changes $(\tilde{v}_{in}(s))$ play the role disturbance.

The schema shown in Fig. 8.54 is called Voltage Mode Control (VMC) since the feedback is taken from output voltage. The feedback can be taken from inductor currents as well. Such a control loop is called Current Mode Control (CMC).

We want to design the controller block shown in the Fig. 8.54. Assume that we want to design a PI controller for this system. Since the control-to-output transfer function has no pole in origin, i.e., $s=0$, the controller must contain an integrator in order to obtain zero steady-state error.

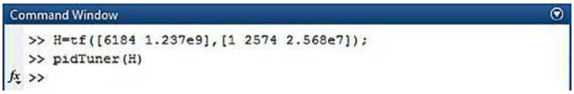
First of all, we must enter the transfer function for plant $\left(\frac{\tilde{v}_o(s)}{\tilde{d}(s)}\right)$ into the MATLAB® environment. This can be done with the aid of `tf` command (Fig. 8.55).



```
>> H=tf([6184 1.237e9],[1 2574 2.568e7]);
fx >>
```

FIG. 8.55 Entering the transfer functions into MATLAB with the aid of `tf` command.

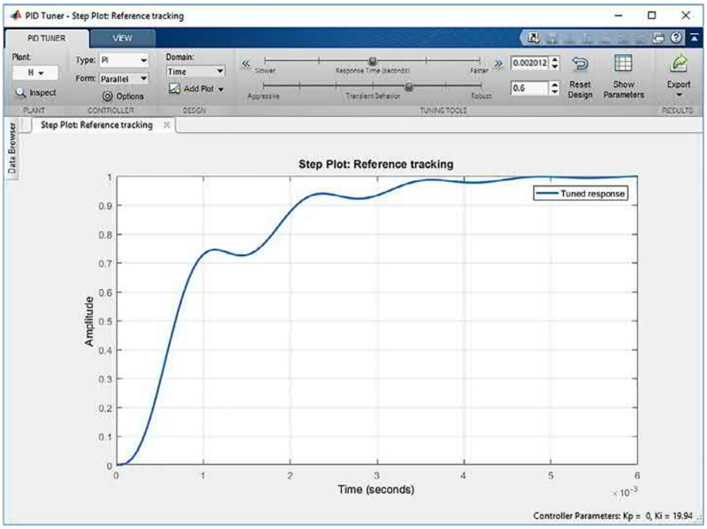
We use the `pidTuner` command to tune the PI controller (Fig. 8.56).



```
>> H=tf([6184 1.237e9],[1 2574 2.568e7]);
>> pidTuner(H)
fx >>
```

FIG. 8.56 The `pidTuner` command is used to tune the PID controller coefficients.

After pressing the Enter key, the window shown in Fig. 8.57 will appear.

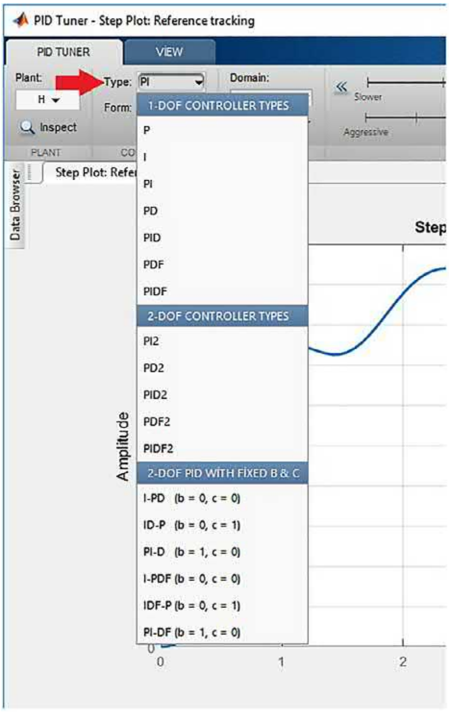


PID Tuner - Step Plot: Reference tracking
PID TUNER
VIEW
Plant:
H
Inspect
Type: PI
Form: Parallel
Options
Domain:
Time
Add Plot
Slower
Response Time (seconds)
Faster
0.002012
Aggressive
Transient Behavior
Robust
0.6
Reset Design
Show Parameters
Export
PLANT
CONTROLLER
DESIGN
TUNING TOOLS
RESULTS
Data Browser
Step Plot: Reference tracking
Step Plot: Reference tracking
Tuned response
Amplitude
1
0.9
0.8
0.7
0.6
0.5
0.4
0.3
0.2
0.1
0
Time (seconds)
0
1
2
3
4
5
6
$\times 10^{-3}$

Controller Parameters: Kp = 0, Ki = 19.94

FIG. 8.57 `pidTuner` application environment.

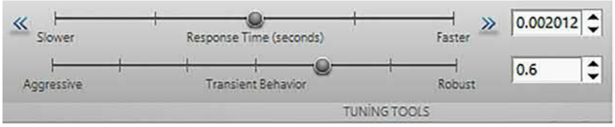
Select desired controller type from the Type drop down list (Fig. 8.58).



PID Tuner - Step Plot: Reference tracking
PID TUNER
VIEW
Plant:
H
Inspect
Type: PI
Form:
Domain:
Slower
Aggressive
PLANT
CO
Data Browser
Step Plot: Refe
Step
Amplitude
1-DOF CONTROLLER TYPES
P
I
PI
PD
PID
PDF
PIDF
2-DOF CONTROLLER TYPES
PI2
PD2
PID2
PDF2
PIDF2
2-DOF PID WITH FIXED B & C
I-PD (b = 0, c = 0)
ID-P (b = 0, c = 1)
PI-D (b = 1, c = 0)
I-PDF (b = 0, c = 0)
IDF-P (b = 0, c = 1)
PI-DF (b = 1, c = 0)
0
0
1
2

FIG. 8.58 Different types of controllers, which are tunable by `pidTuner` application.

Use the sliders to obtain the desired time response (Fig. 8.59).

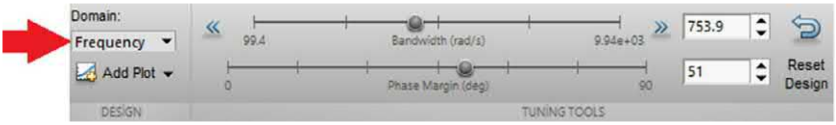

Slower
Faster
Aggressive
Robust
TUNING TOOLS

Response Time (seconds) 0.002012

Transient Behavior 0.6

FIG. 8.59 Sliders are used to obtain the desired response.

You can use the Domain: Frequency (Fig. 8.60) to tune the controller using frequency domain criteria, i.e., bandwidth and phase margin. Generally, phase margin must be greater than 45°.


99.4
9.94e+03
0
90
Add Plot
DESIGN
TUNING TOOLS
Reset Design

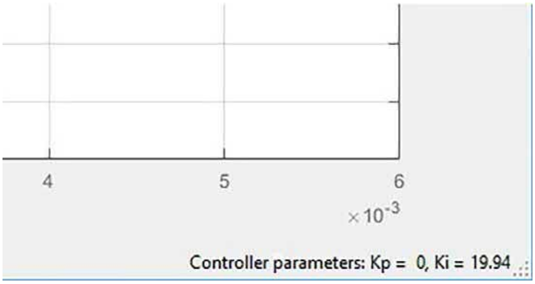
Domain: Frequency

Bandwidth (rad/s) 753.9

Phase Margin (deg) 51

FIG. 8.60 Tuning the controller in frequency domain.

The PID Tuner shows the value of gains in the right bottom of the window (Kp is the proportional gain and Ki is integrator gain) (Fig. 8.61).



Controller parameters: Kp = 0, Ki = 19.94

FIG. 8.61 Kp and Ki show the value of proportional gain and integral gain (for a PI controller), respectively.

You can export the designed controller to MATLAB Workspace by clicking the Export button (Fig. 8.62).

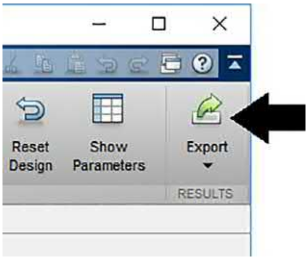

Reset Design
Show Parameters
Export
RESULTS

FIG. 8.62 Transferring the designed controller into MATLAB® Workspace.

8.19 Input/output impedance of converter

The input impedance of a DC-DC converter is the impedance seen from the input DC source. The output impedance is defined as the output voltage response of converter for the excitation of current i_Z at constant input voltage V_G and duty ratio D. In some descriptions, the output impedance includes the load, in others it does not (Fig. 8.63).

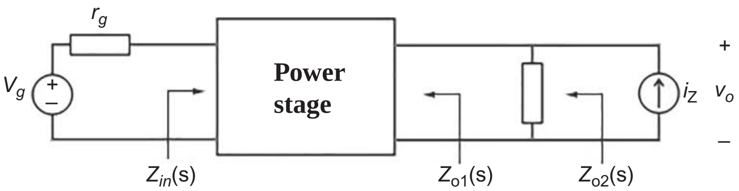


FIG. 8.63 Input impedance of converter and two variants of the output impedance of converter.

The input/output of converter can be extracted in the same way that control-to-output transfer function ($\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$) extracted. Assume that we want to extract the input and output impedance of the buck converter shown in Fig. 8.64.

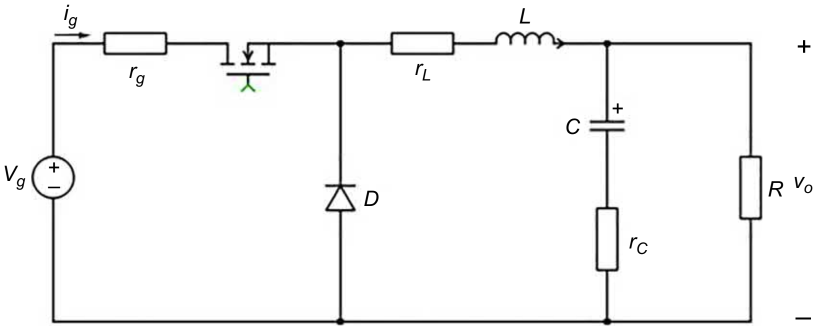


FIG. 8.64 Schematic of a buck converter. rg, r_L, and r_C show the internal resistance of input source, inductor ESR, and capacitor ESR, respectively.

As earlier when the MOSFET switch is closed, the diode is reverse-biased. The equivalent circuit of Fig. 8.65 represents this case. io is a fictitious current source added to the circuit in order to measure the output impedance ($Z_o(s) = \frac{\text{vo(s)}}{\text{io(s)}}$).

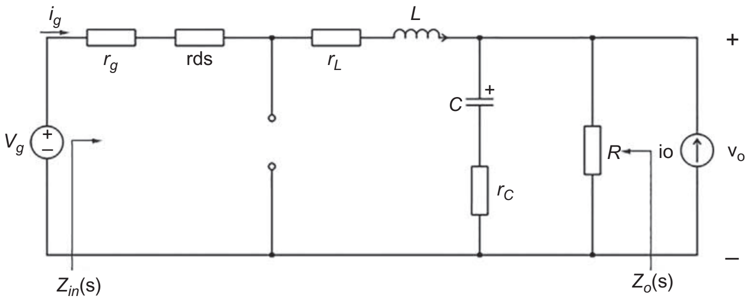


FIG. 8.65 Equivalent circuit of buck converter with closed MOSFET.

According to Fig. 8.65, the circuit differential equations can be written as:

$$\begin{cases} \dfrac{di_L(t)}{dt} = \dfrac{1}{L}\left(-\left(r_g + r_{ds} + r_L + \dfrac{R\times r_C}{R+r_C}\right)i_L - \dfrac{R}{R+r_C}v_C - \dfrac{R\times r_C}{R+r_C}i_O + v_g\right) \\ \dfrac{dv_C(t)}{dt} = \dfrac{1}{C}\left(\dfrac{R}{R+r_C}i_L - \dfrac{1}{R+r_C}v_C + \dfrac{R}{R+r_C}i_O\right) \end{cases} \tag{8.141}$$

$$i_g = i_L$$

$$v_o = r_C C\frac{dv_C}{dt} + v_C = \frac{R\times r_C}{R+r_C}i_L + \frac{R}{R+r_C}v_C + \frac{R\times r_C}{R+r_C}i_O$$

We need to define two outputs (i_g and v_o) in order to calculate the input/output impedance. Applying the SSA to the equations leads to six different transfer functions: $\frac{\tilde{i}_g(s)}{\tilde{d}(s)}$, $\frac{\tilde{i}_g(s)}{\tilde{v}_g(s)}$, $\frac{\tilde{i}_g(s)}{\tilde{i}_o(s)}$, $\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$, $\frac{\tilde{v}_o(s)}{\tilde{v}_g(s)}$, and $\frac{\tilde{v}_o(s)}{\tilde{i}_o(s)}$. Output impedance of the converter is extracted with the aid of $\frac{\tilde{v}_o(s)}{\tilde{i}_o(s)}$. The input impedance of converter is calculated using $\frac{1}{\frac{\tilde{i}_g(s)}{\tilde{v}_g(s)}}$.

When the MOSFET is open, the diode becomes forward-biased to carry the inductor current. The equivalent circuit shown in Fig. 8.66 represents this case. As before, rD and VD show the diode resistance and diode forward voltage drop, respectively.

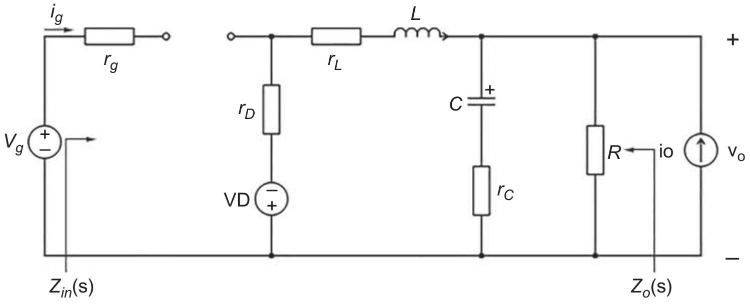


FIG. 8.66 Equivalent circuit of buck converter with open MOSFET.

According to Fig. 8.66, the circuit differential equations can be written as:

$$\begin{cases} \dfrac{di_L(t)}{dt} = \dfrac{1}{L}\left(-\left(r_D + r_L + \dfrac{R\times r_C}{R+r_C}\right)i_L - \dfrac{R}{R+r_C}v_C - \dfrac{R\times r_C}{R+r_C}i_O - v_D\right) \\ \dfrac{dv_C(t)}{dt} = \dfrac{1}{C}\left(\dfrac{R}{R+r_C}i_L - \dfrac{1}{R+r_C}v_C + \dfrac{R}{R+r_C}i_O\right) \end{cases} \tag{8.142}$$

$$i_g = 0$$

$$v_o = r_C C\frac{dv_C}{dt} + v_C = \frac{R\times r_C}{R+r_C}i_L + \frac{R}{R+r_C}v_C + \frac{R\times r_C}{R+r_C}i_O$$

Consider a buck converter with parameters as shown in Table 8.3.

TABLE 8.3 The buck converter parameters (see Fig. 8.64).

	Nominal Value
Output voltage, vo	20 V
Duty ratio, D	0.4
Input DC source voltage, Vg	50 V
Input DC source internal resistance, rg	0.01 Ω
MOSFET Drain-Source resistance, rds	40 mΩ
Capacitor, C	100 μF
Capacitor ESR, rC	0.05 Ω
Inductor, L	400 μH
Inductor ESR, rL	50 mΩ
Diode voltage drop, vD	0.7 V
Diode forward resistance, rD	10 mΩ
Load resistor, R	20 Ω
Switching Frequency, Fsw	20 kHz

The following program extracts the input and output impedance of buck converter with the parameters as shown in Table 8.3. Note that variable `vo_io` keeps the $\frac{\tilde{v}_o(s)}{\tilde{i}_o(s)}$ transfer function, variable `vo_d` keeps the $\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$ transfer function, so on.

```
%This program calculates the input and output impedance of the Buck
%converter.

clc

clear all
syms vg rg d rL L rC C R vC iL rds rD vD io

%Converter Dynamical equations
%M1: diL/dt for closed MOSFET.
%M2: dvC/dt for closed MOSFET.
%M3: current of input DC source for closed MOSFET.
%M4: output voltage of converter for closed MOSFET.

%M5: diL/dt for open MOSFET.
%M6: dvC/dt for open MOSFET.
%M7: current of input DC source for open MOSFET.
%M8: output voltage of converter for open MOSFET.

M1=(-(rg+rds+rL+R*rC/(R+rC))*iL-R/(R+rC)*vC-R*rC/(R+rC)*
io+vg)/L;
M2=(R/(R+rC)*iL-1/(R+rC)*vC+R/(R+rC)*io)/C;
M3=iL;
M4=R*rC/(R+rC)*iL+R/(R+rC)*vC+R*rC/(R+rC)*io;

M5=(-(rD+rL+R*rC/(R+rC))*iL-R/(R+rC)*vC-R*rC/(R+rC)*io-vD)/L;
M6=(R/(R+rC)*iL-1/(R+rC)*vC+R/(R+rC)*io)/C;
M7=0;
M8=R*rC/(R+rC)*iL+R/(R+rC)*vC+R*rC/(R+rC)*io;

%Averaged Equations
diL_dt_ave=simplify(M1*d+M5*(1-d));
dvC_dt_ave=simplify(M2*d+M6*(1-d));
ig_ave=simplify(M3*d+M7*(1-d));
vo_ave=simplify(M4*d+M8*(1-d));

%DC Operating Point
DC=solve(diL_dt_ave==0,dvC_dt_ave==0,'iL','vC');
IL=DC.iL;
VC=DC.vC;

%Linearization
A11=simplify(subs(diff(diL_dt_ave,iL),[iL vC io],[IL VC 0]));
A12=simplify(subs(diff(diL_dt_ave,vC),[iL vC io],[IL VC 0]));
A21=simplify(subs(diff(dvC_dt_ave,iL),[iL vC io],[IL VC 0]));
A22=simplify(subs(diff(dvC_dt_ave,vC),[iL vC io],[IL VC 0]));
AA=[A11 A12;A21 A22];

B11=simplify(subs(diff(diL_dt_ave,io),[iL vC io],[IL VC 0]));
B12=simplify(subs(diff(diL_dt_ave,vg),[iL vC io],[IL VC 0]));
B13=simplify(subs(diff(diL_dt_ave,d),[iL vC io],[IL VC 0]));
```

```
B21=simplify(subs(diff(dvC_dt_ave,io),[iL vC io],[IL VC 0]));
B22=simplify(subs(diff(dvC_dt_ave,vg),[iL vC io],[IL VC 0]));
B23=simplify(subs(diff(dvC_dt_ave,d),[iL vC io],[IL VC 0]));

BB=[B11 B12 B13;B21 B22 B23];

C11=simplify(subs(diff(ig_ave,iL),[iL vC io],[IL VC 0]));
C12=simplify(subs(diff(ig_ave,vC),[iL vC io],[IL VC 0]));

C21=simplify(subs(diff(vo_ave,iL),[iL vC io],[IL VC 0]));
C22=simplify(subs(diff(vo_ave,vC),[iL vC io],[IL VC 0]));
CC=[C11 C12; C21 C22];

D11=simplify(subs(diff(ig_ave,io),[iL vC io],[IL VC 0 ]));
D12=simplify(subs(diff(ig_ave,vg),[iL vC io],[IL VC 0]));
D13=simplify(subs(diff(ig_ave,d),[iL vC io],[IL VC 0]));

D21=simplify(subs(diff(vo_ave,io),[iL vC io],[IL VC 0 ]));
D22=simplify(subs(diff(vo_ave,vg),[iL vC io],[IL VC 0]));
D23=simplify(subs(diff(vo_ave,d),[iL vC io],[IL VC 0]));
DD=[D11 D12 D13;D21 D22 D23];

%Components Values
%Variables have underline are used to store the numeric values of
components
%Variables without underline are symbolic variables.
%for example:
%L: symbolic vvariable shows the inductor inductance
%L_: numeric variable shows the inductor inductance value.
L_=400e-6;
rL_=.05;
C_=100e-6;
rC_=.05;
rds_=.04;
rD_=.01;
VD_=.7;
D_=.4;
VG_=50;
rg_=.01;
R_=20;

AA_=eval(subs(AA,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
BB_=eval(subs(BB,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
CC_=eval(subs(CC,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
```

```
DD_=eval(subs(DD,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));

sys=ss(AA_,BB_,CC_,DD_);
sys.stateName={'iL','vC'};
sys.inputname={'io','vg','d'};
sys.outputname={'ig','vo'};

ig_io=sys(1,1);
ig_vg=sys(1,2);
ig_d=sys(1,3);

vo_io=sys(2,1);
vo_vg=sys(2,2);
vo_d=sys(2,3);

Zin=1/ig_vg; %input impedance
Zout=vo_io; %output impedance

%Draws the bode diagram of input/output impedance
figure(1)
bode(Zin), grid minor

figure(2)
bode(Zout), grid minor

%Display the DC operating point of converter
disp('steady state operating point of converter')
disp('IL')
disp(eval(subs(IL,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0])));
disp('VC')
disp(eval(subs(VC,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0])));
```

The program gives the following transfer functions:

$$
\begin{aligned}
&\frac{\tilde{v}_o(s)}{\tilde{d}(s)}=6316.8\frac{s+2\times10^5}{s^2+813.4s+2.503\times10^7}\\
&Z_{in}(s)=\frac{\tilde{v}_g(s)}{\tilde{i}_g(s)}=0.0025\frac{s^2+813.4s+2.503\times10^7}{s+498.8}\\
&Z_o(s)=\frac{\tilde{v}_o(s)}{\tilde{i}_o(s)}=0.049875\frac{\left(s+2\times10^5\right)(s+190)}{s^2+813.4s+2.503\times10^7}\\
&\frac{\tilde{v}_o(s)}{\tilde{v}_g(s)}=49.875\frac{\left(s+2\times10^5\right)}{s^2+813.4s+2.503\times10^7}
\end{aligned}
\tag{8.143}
$$

Bode diagram of control-to-output transfer function ($\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$), input impedance ($Z_{in}(s)=\frac{\tilde{v}_g(s)}{\tilde{i}_g}(s)$), output impedance ($Z_o(s)=\frac{\tilde{v}_o(s)}{\tilde{i}_o}(s)$), and audio susceptibility ($\frac{\tilde{v}_o(s)}{\tilde{v}_g(s)}$) are shown in Figs. 8.67, 8.68, 8.69, and 8.70, respectively.

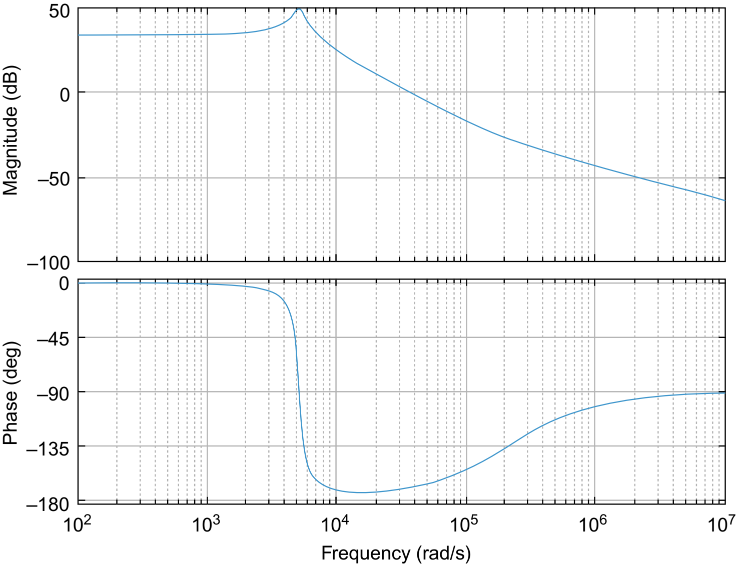


FIG. 8.67 Bode diagram of control-to-output transfer function ($\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$) for the buck converter with parameters as shown in Table 8.3.

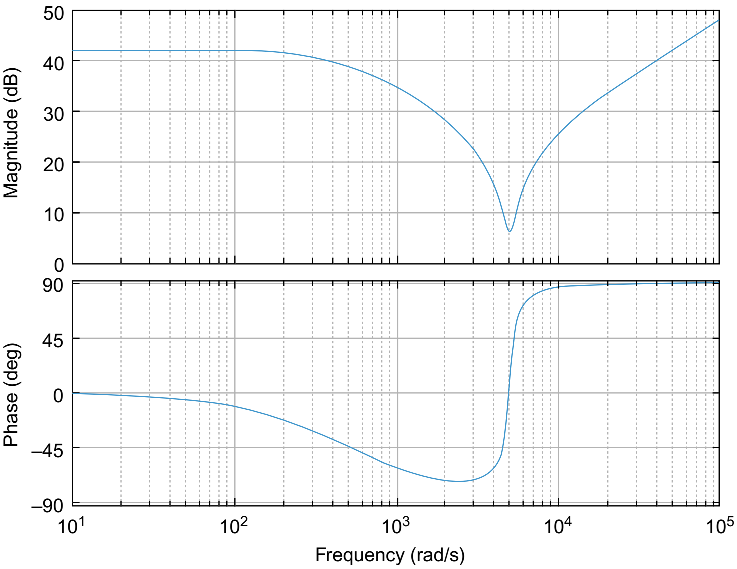


FIG. 8.68 Bode diagram of input impedance ($Z_{in}(s)$) for the buck converter with parameters as shown in Table 8.3.

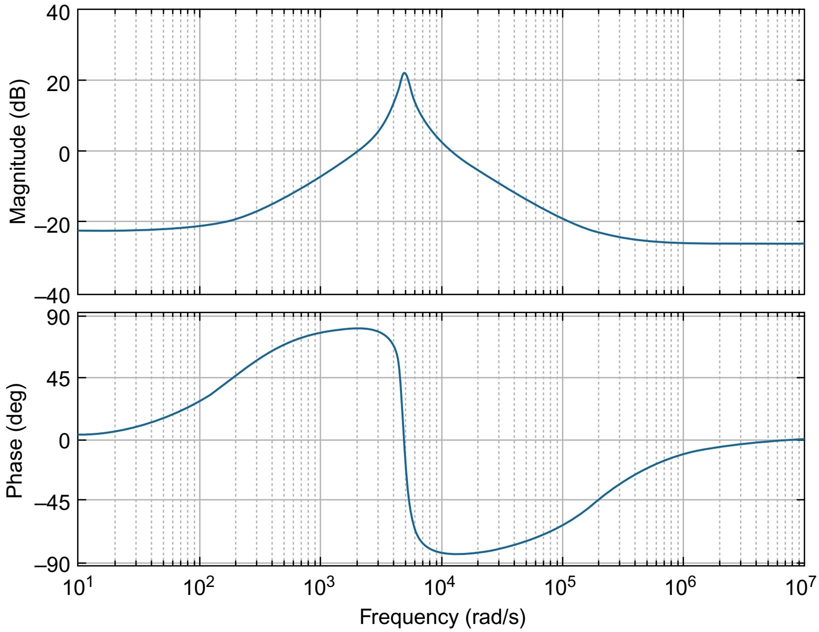


FIG. 8.69 Bode diagram of output impedance ($Z_o(s)$) for the buck converter with parameters as shown in Table 8.3.

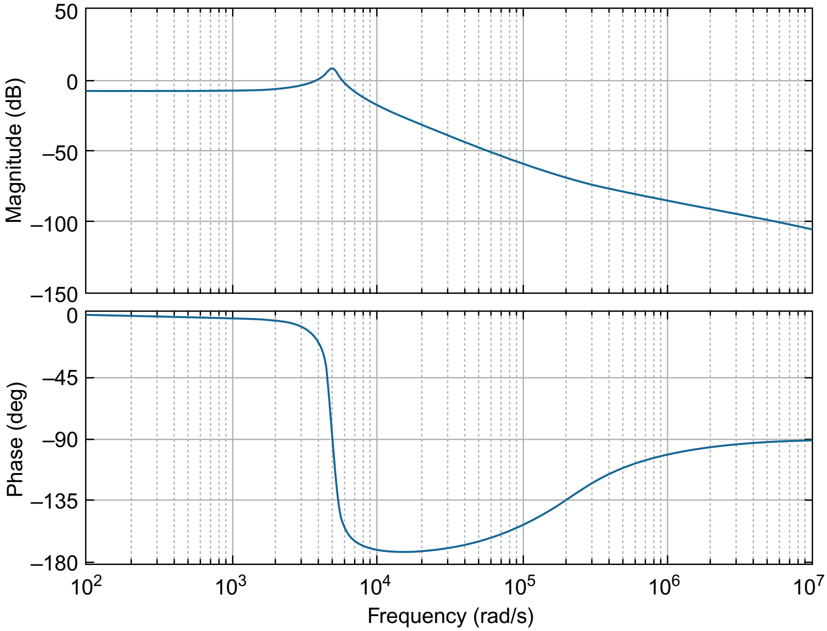


FIG. 8.70 Bode diagram of audio susceptibility ($\frac{\tilde{v}_o(s)}{\tilde{v}_g(s)}$) for the buck converter with parameters as shown in Table 8.3.

The dynamical model shown in Fig. 8.71 can be drawn for the buck converter with parameters as shown in Table 8.3.

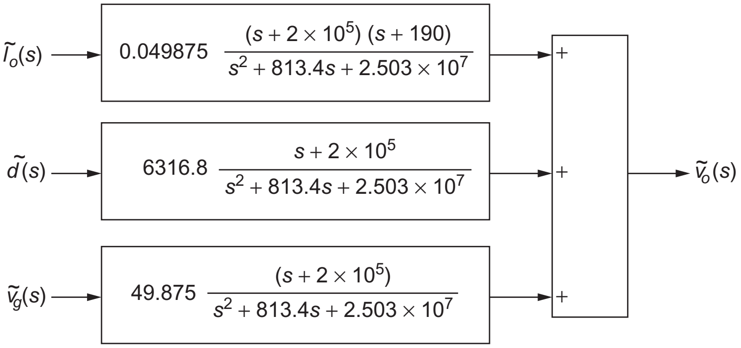

$\tilde{i}_o(s)$
$0.049875 \frac{(s+2\times10^5)(s+190)}{s^2+813.4s+2.503\times10^7}$
$\tilde{d}(s)$
$6316.8 \frac{s+2\times10^5}{s^2+813.4s+2.503\times10^7}$
$\tilde{v}_g(s)$
$49.875 \frac{(s+2\times10^5)}{s^2+813.4s+2.503\times10^7}$
+
+
+
$\tilde{v}_o(s)$

FIG. 8.71 Dynamical model of the studied buck converter.

8.20 Effect of feedback control on output impedance

Fig. 8.71 shows the open loop model of buck converter with the parameters shown in Table 8.3. Consider a simple feedback control system such as the one shown in Fig. 8.72.

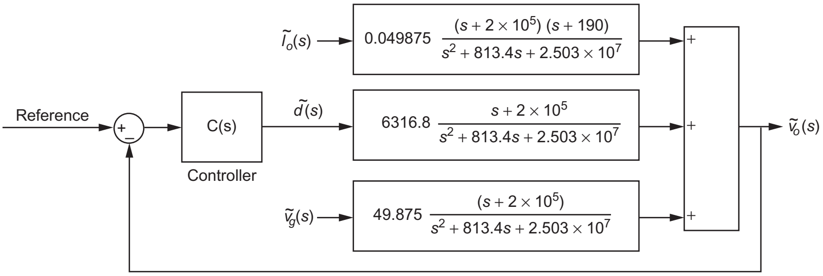

Reference
+
−
C(s)
Controller
$\tilde{i}_o(s)$
$0.049875 \frac{(s+2\times10^5)(s+190)}{s^2+813.4s+2.503\times10^7}$
$\tilde{d}(s)$
$6316.8 \frac{s+2\times10^5}{s^2+813.4s+2.503\times10^7}$
$\tilde{v}_g(s)$
$49.875 \frac{(s+2\times10^5)}{s^2+813.4s+2.503\times10^7}$
+
+
+
$\tilde{v}_o(s)$

FIG. 8.72 Voltage Mode (VM) control of the studied buck converter. The $\tilde{i}_o$ and $\tilde{v}_g$ inputs play the role of disturbance.

Assume that the controller is a simple I-type controller ($C(s)=\frac{4.85}{s}$). Fig. 8.73 shows the step response (from Reference input to $\tilde{v}_o$ output) of the closed loop.

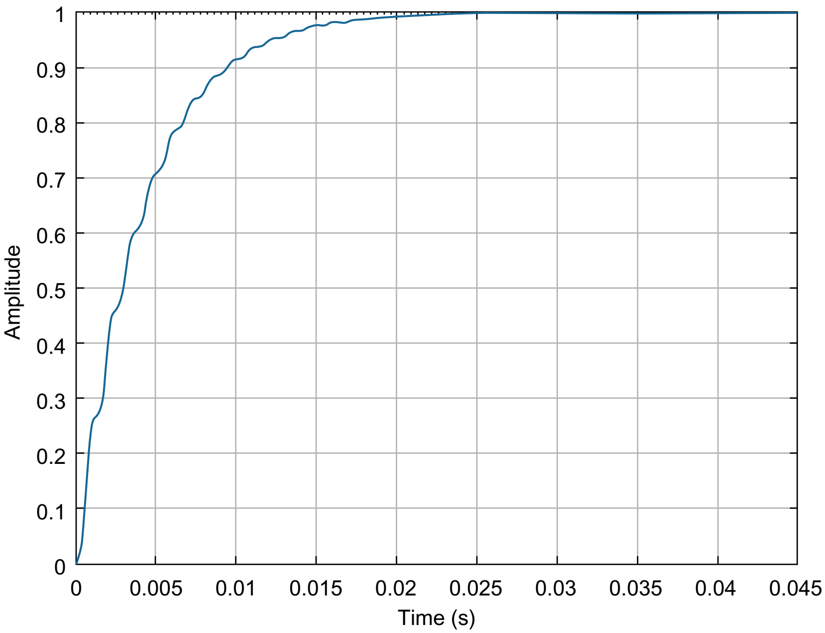


FIG. 8.73 Step response of closed-loop control system shown in Fig. 8.72 with $C(s) = \frac{4.85}{s}$.

According to Fig. 8.72, the closed-loop output impedance ($Z_{o,CL}(s)$) is:

$$Z_{o,\ CL}(s) = Z_{o,\ OL}(s) \times \frac{1}{1 + C(s) \times \dfrac{vo(s)}{d(s)}} \tag{8.144}$$

$$\begin{aligned} Z_{o,\ CL}(s) &= 0.049875 \times \frac{\left(s + 2 \times 10^5\right)(s + 190)}{s^2 + 813.4s + 2.503 \times 10^7} \\ &\quad \times \frac{1}{1 + \dfrac{4.85}{s} \times 6316.8 \times \dfrac{\left(s + 2 \times 10^5\right)}{s^2 + 813.4s + 2.503 \times 10^7}} \\ &= \frac{0.04988s^5 + 10^4 s^4 + 1.13 \times 10^7 s^3 + 2.515 \times 10^{11} s^2 + 4.744 \times 10^{13} s}{s^5 + 1627s^4 + 5.075 \times 10^7 s^3 + 4.687 \times 10^{10} + 6.323 \times 10^{14} s + 1.534 \times 10^{17}} \end{aligned} \tag{8.145}$$

In order to calculate the closed-loop output impedance, set the $\tilde{v}_g$ and Reference input to zero and calculate the transfer function from $\tilde{i}_o$ input to $\tilde{v}_o$ output.

Fig. 8.74 is a comparison between the open-loop output impedance ($Z_{o,OL}(s)$, Eq. (8.143)) and the closed-loop output impedance ($Z_{o,CL}(s)$, Eq. (8.145)).

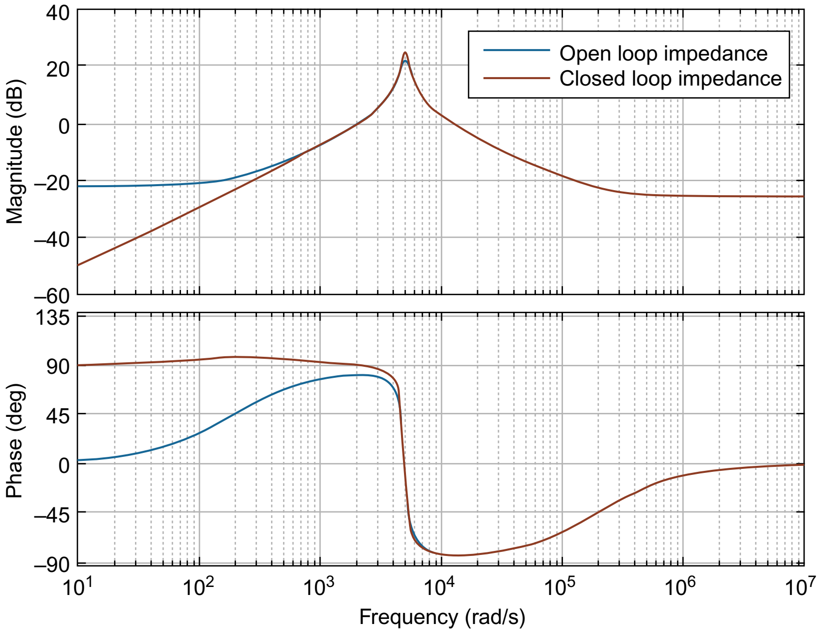


FIG. 8.74 Comparison between open-loop output impedance and closed-loop output impedance. Feedback reduced the output impedance.

The closed-loop output impedance is reduced at low-frequency portion of the graph. Reduction of output impedance is one of the desired properties of feedback control.

8.21 Dynamic of buck-boost converter

Schematic of buck-boost converter is shown in Fig. 8.75.

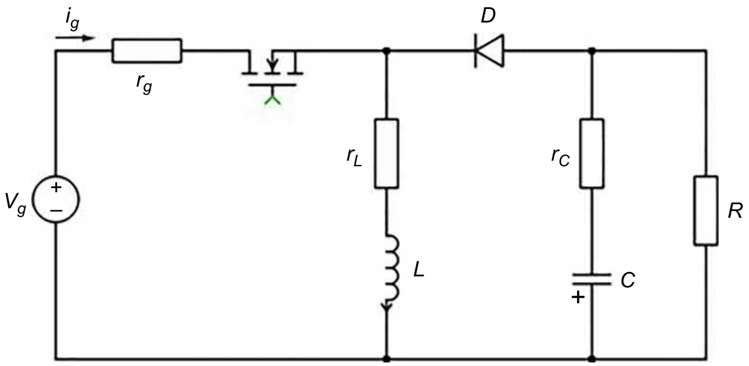


FIG. 8.75 Schematic of buck-boost converter.

When the MOSFET is closed, the diode is reverse biased. According to Fig. 8.76, the circuit differential equation is:

$$\begin{cases} \dfrac{di_L(t)}{dt} = \dfrac{1}{L}\left(-\left(r_g + r_{ds} + r_L\right)i_L + v_g\right) \\ \dfrac{dv_C(t)}{dt} = \dfrac{1}{C}\left(\dfrac{R}{R+r_C}i_O - \dfrac{1}{R+r_C}v_C\right) \end{cases}$$
$$i_g = i_L \qquad (8.146)$$
$$v_o = \frac{R}{R+r_C}v_C + \frac{R \times r_C}{R+r_C}i_O$$

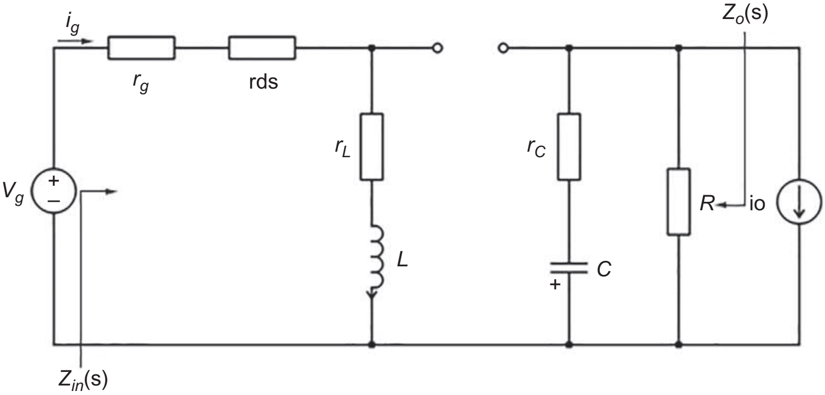


FIG. 8.76 Equivalent circuit of buck-boost converter with closed MOSFET.

When the MOSFET switch is opened, the diode becomes forward-biased. According to Fig. 8.77, the circuit differential equation is:

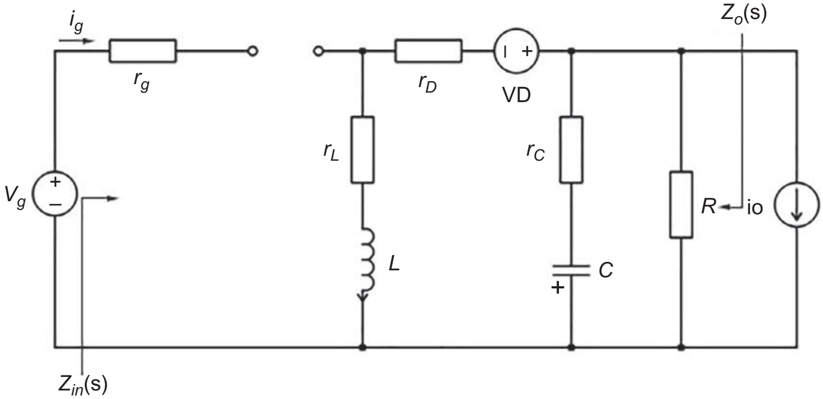


FIG. 8.77 Equivalent circuit of buck-boost converter with open MOSFET.

$$
\begin{aligned}
&\begin{cases}
\dfrac{di_L(t)}{dt} = \dfrac{1}{L}\left(-\left(r_D + r_L + \dfrac{R \times r_C}{R + r_C}\right)i_L - \dfrac{R}{R + r_C}v_C - \dfrac{R \times r_C}{R + r_C}i_O - v_D\right)\\
\dfrac{dv_C(t)}{dt} = \dfrac{1}{C}\left(\dfrac{R}{R + r_C}i_L - \dfrac{1}{R + r_C}v_C + \dfrac{R}{R + r_C}i_O\right)
\end{cases}\\
&i_g = 0\\
&v_o = \dfrac{R \times r_C}{R + r_C}i_L + \dfrac{R}{R + r_C}v_C + \dfrac{R \times r_C}{R + r_C}i_O + \mathrm{V_D}
\end{aligned}
\tag{8.147}
$$

Assume a buck-boost converter with the following parameters.

The following program extracts the small-signal transfer functions of a buck-boost converter with component values as shown in Table 8.4.

TABLE 8.4 The buck-boost converter parameters (see Fig. 8.75).

	Nominal value
Output voltage, vo	−16 V
Duty ratio, D	0.4
Input DC source voltage, Vg	24 V
Input DC source internal resistance, rg	0.1 Ω
MOSFET Drain-Source resistance, rds	40 mΩ
Capacitor, C	80 μF
Capacitor ESR, rC	0.05 Ω
Inductor, L	20 μH
Inductor ESR, rL	10 mΩ
Diode voltage drop, vD	0.7 V
Diode forward resistance, rD	10 mΩ
Load resistor, R	5 Ω
Switching Frequency, Fsw	100 kHz

```
%This program calculates the small signal transfer functions of
Buck-Boost
%converter.

clc
```

```
clear all
syms vg rg d rL L rC C R vC iL rds rD vD io

%Converter Dynamical equations
%M1: diL/dt for closed MOSFET.
%M2: dvC/dt for closed MOSFET.
%M3: current of input DC source for closed MOSFET.
%M4: output voltage of converter for closed MOSFET.

%M5: diL/dt for open MOSFET.
%M6: dvC/dt for open MOSFET.
%M7: current of input DC source for open MOSFET.
%M8: output voltage of converter for open MOSFET.

M1=(-(rg+rds+rL)*iL+vg)/L;
M2=(R/(R+rC)*io-vC/(R+rC))/C;
M3=iL;
M4=R*rC/(R+rC)*io+R/(R+rC)*vC;

M5=(-(rL+rD+rC*R/(R+rC))*iL-R/(R+rC)*vC-R*rC/(R+rC)*io-vD)/L;
M6=(R/(R+rC)*iL-1/(R+rC)*vC+R/(R+rC)*io)/C;
M7=0;
M8=rC*R/(rC+R)*iL+R/(R+rC)*vC+R*rC/(R+rC)*io+vD;

%Averaged Equations
diL_dt_ave=simplify(M1*d+M5*(1-d));
dvC_dt_ave=simplify(M2*d+M6*(1-d));
ig_ave=simplify(M3*d+M7*(1-d));
vo_ave=simplify(M4*d+M8*(1-d));

%DC Operating Point
DC=solve(diL_dt_ave==0,dvC_dt_ave==0,'iL','vC');
IL=DC.iL;
VC=DC.vC;

%Linearization
A11=simplify(subs(diff(diL_dt_ave,iL),[iL vC io],[IL VC 0]));
A12=simplify(subs(diff(diL_dt_ave,vC),[iL vC io],[IL VC 0]));
A21=simplify(subs(diff(dvC_dt_ave,iL),[iL vC io],[IL VC 0]));
A22=simplify(subs(diff(dvC_dt_ave,vC),[iL vC io],[IL VC 0]));
AA=[A11 A12;A21 A22];

B11=simplify(subs(diff(diL_dt_ave,io),[iL vC io],[IL VC 0]));
B12=simplify(subs(diff(diL_dt_ave,vg),[iL vC io],[IL VC 0]));
B13=simplify(subs(diff(diL_dt_ave,d),[iL vC io],[IL VC 0]));

B21=simplify(subs(diff(dvC_dt_ave,io),[iL vC io],[IL VC 0]));
B22=simplify(subs(diff(dvC_dt_ave,vg),[iL vC io],[IL VC 0]));
B23=simplify(subs(diff(dvC_dt_ave,d),[iL vC io],[IL VC 0]));
```

```
BB=[B11 B12 B13;B21 B22 B23];

C11=simplify(subs(diff(ig_ave,iL),[iL vC io],[IL VC 0]));
C12=simplify(subs(diff(ig_ave,vC),[iL vC io],[IL VC 0]));

C21=simplify(subs(diff(vo_ave,iL),[iL vC io],[IL VC 0]));
C22=simplify(subs(diff(vo_ave,vC),[iL vC io],[IL VC 0]));
CC=[C11 C12; C21 C22];

D11=simplify(subs(diff(ig_ave,io),[iL vC io],[IL VC 0 ]));
D12=simplify(subs(diff(ig_ave,vg),[iL vC io],[IL VC 0]));
D13=simplify(subs(diff(ig_ave,d),[iL vC io],[IL VC 0]));

D21=simplify(subs(diff(vo_ave,io),[iL vC io],[IL VC 0 ]));
D22=simplify(subs(diff(vo_ave,vg),[iL vC io],[IL VC 0]));
D23=simplify(subs(diff(vo_ave,d),[iL vC io],[IL VC 0]));
DD=[D11 D12 D13;D21 D22 D23];

%Components Values
%Variables have underline are used to store the numeric values of
components
%Variables without underline are symbolic variables.
%for example:
%L: symbolic vvariable shows the inductor inductance
%L_: numeric variable shows the inductor inductance value.
L_=20e-6;
rL_=.01;
C_=80e-6;
rC_=.05;
rds_=.04;
rD_=.01;
VD_=.7;
D_=.4;
VG_=24;
rg_=.1;
R_=5;

AA_=eval(subs(AA,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
BB_=eval(subs(BB,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
CC_=eval(subs(CC,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
DD_=eval(subs(DD,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));

sys=ss(AA_,BB_,CC_,DD_);
sys.stateName={'iL','vC'};
sys.inputname={'io','vg','d'};
sys.outputname={'ig','vo'};

ig_io=sys(1,1);
ig_vg=sys(1,2);
ig_d=sys(1,3);

vo_io=sys(2,1);
vo_vg=sys(2,2);
vo_d=sys(2,3);

Zin=1/ig_vg; %input impedance
Zout=vo_io; %output impedance

%Draws the bode diagram of input/output impedance
figure(1)
bode(Zin), grid minor

figure(2)
bode(Zout), grid minor

%Display the DC operating point of converter
disp('steady state operating point of converter')
disp('IL')
disp(eval(subs(IL,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0])));
disp('VC')
disp(eval(subs(VC,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0])));
```

The program gives the following results:

$$\begin{aligned}
\frac{v_o(s)}{d(s)} &= -0.94123\frac{\left(s+1.267\times10^5\right)\left(s-1.168\times10^5\right)}{s^2+7560s+2.332\times10^8}\\
Z_{in}(s) = \frac{v_g(s)}{i_g(s)} &= 0.000125\frac{s^2+7560s+2.332\times10^8}{s+2475}\\
Z_o(s) = \frac{v_o(s)}{i_o(s)} &= 0.049505\frac{\left(s+2.5\times10^5\right)(s+4194)}{s^2+7560s+2.332\times10^8}
\end{aligned} \tag{8.148}$$

Note that the obtained control-to-output transfer function is nonminimum phase, i.e., has a zero in right half-plane. Bode diagram of control-to-output transfer function, open loop input impedance, and open loop output impedance are shown in Figs. 8.78, 8.79, and 8.80, respectively.

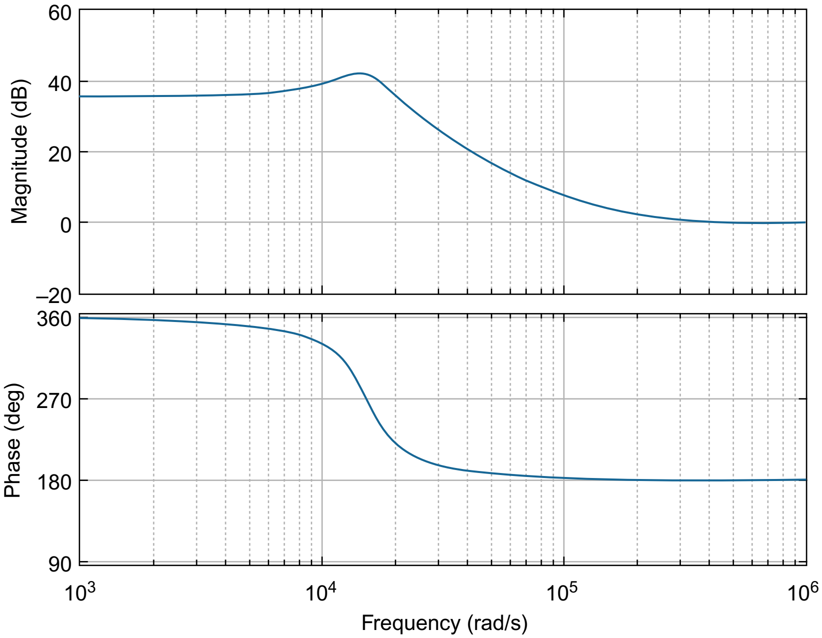


FIG. 8.78 Bode diagram of control-to-output transfer function ($\frac{v_o(s)}{d(s)}$) for the buck-boost converter with parameters as shown in Table 8.4.

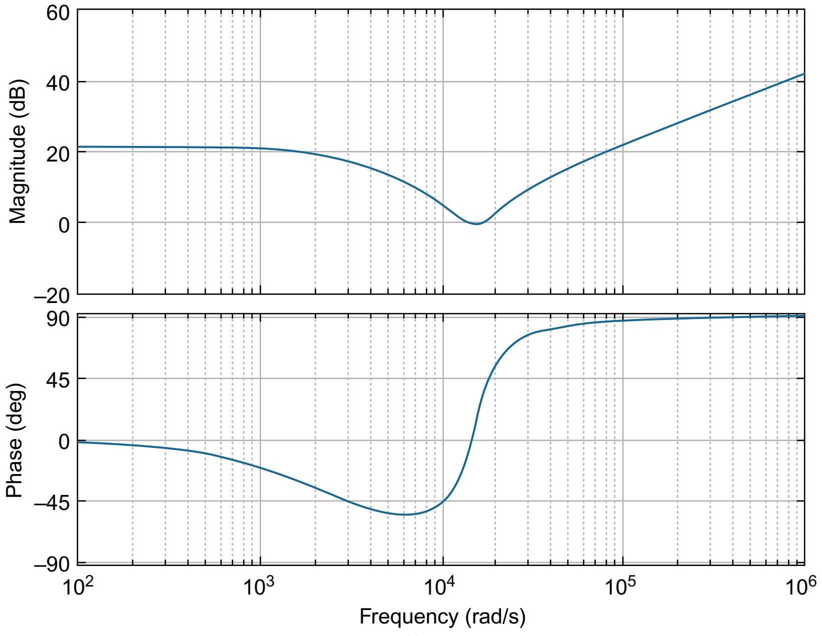


FIG. 8.79 Bode diagram of input impedance ($\frac{v_g(s)}{i_g(s)}$) for the buck-boost converter with parameters as shown in Table 8.4.

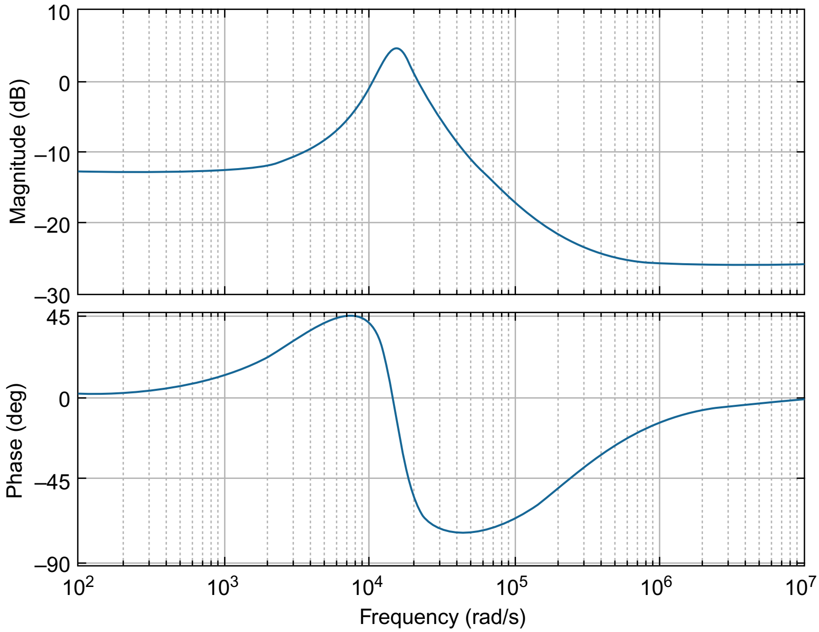


FIG. 8.80 Bode diagram of output impedance $\left(\frac{\tilde{v}_o(s)}{\tilde{i}_o(s)}\right)$ for the buck-boost converter with parameters as shown in Table 8.4.

8.22 Dynamics of boost converter

Schematic of boost converter is shown in Fig. 8.81.

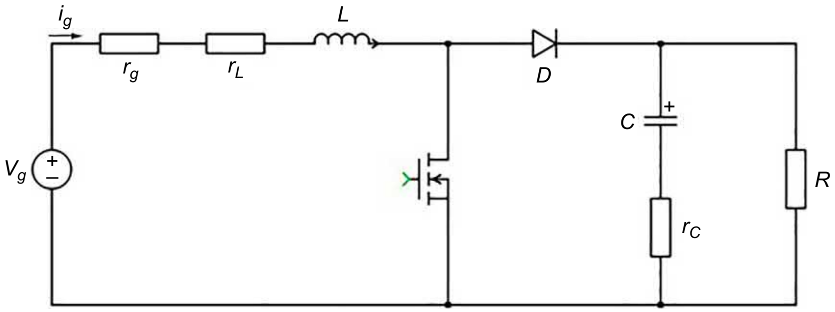


FIG. 8.81 Schematic of boost converter.

When the MOSFET is closed, the diode is reverse biased.

According to Fig. 8.82, the circuit differential equation is:

$$\begin{cases} \dfrac{di_L(t)}{dt} = \dfrac{1}{L}\left(-\left(r_g + r_{ds} + r_L\right)i_L + v_g\right) \\ \dfrac{dv_C(t)}{dt} = \dfrac{1}{C}\left(-\dfrac{1}{R+r_C}v_C + \dfrac{R}{R+r_C}i_O\right) \end{cases} \tag{8.149}$$

$$i_g = i_L$$

$$v_o = \frac{R}{R+r_C}v_C + \frac{R\times r_C}{R+r_C}i_O$$

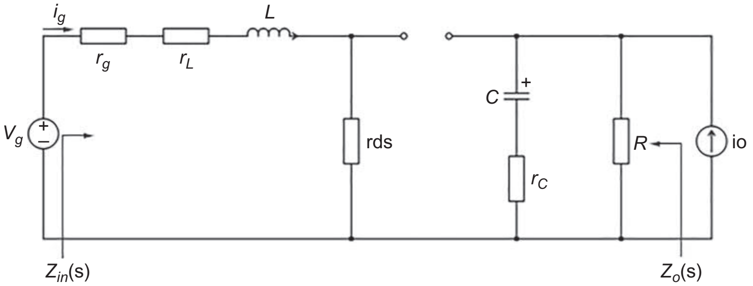


FIG. 8.82 Equivalent circuit of boost converter with closed MOSFET.

When the MOSFET switch is opened, the diode becomes forward-biased. According to Fig. 8.83, the circuit differential equation is:

$$\begin{cases} \dfrac{di_L(t)}{dt} = \dfrac{1}{L}\left(-\left(r_g + r_L + r_D + \dfrac{R\times r_C}{R+r_C}\right)i_L - \dfrac{R}{R+r_C}v_C - \dfrac{R\times r_C}{R+r_C}i_O + v_g - v_D\right) \\ \dfrac{dv_C(t)}{dt} = \dfrac{1}{C}\left(\dfrac{R}{R+r_C}i_L - \dfrac{1}{R+r_C}v_C + \dfrac{R}{R+r_C}i_O\right) \end{cases}$$

$$i_g = i_L$$

$$v_o = \frac{R\times r_C}{R+r_C}i_L + \frac{R}{R+r_C}v_C + \frac{R\times r_C}{R+r_C}i_O \tag{8.150}$$

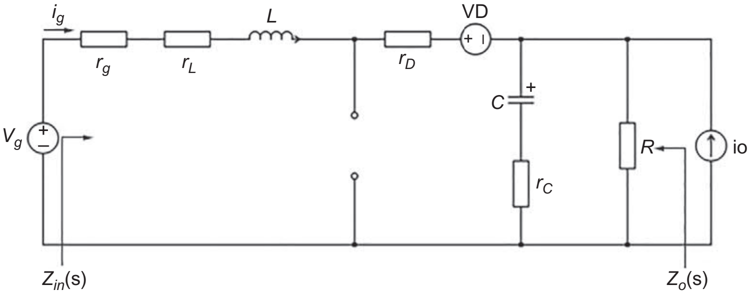


FIG. 8.83 Equivalent circuit of boost converter with open MOSFET.

Assume a boost converter with the parameters as shown in Table 8.5.

TABLE 8.5 **The boost converter parameters (see Fig. 8.81).**

	Nominal value
Output voltage, vo	30 V
Duty ratio, D	0.6
Input DC source voltage, Vg	12 V
Input DC source internal resistance, rg	0.1 Ω
MOSFET Drain-Source resistance, rds	40 mΩ
Capacitor, C	100 µF
Capacitor ESR, rC	0.05 Ω
Inductor, L	120 µH
Inductor ESR, rL	10 mΩ
Diode voltage drop, vD	0.7 V
Diode forward resistance, rD	10 mΩ
Load resistor, R	50 Ω
Switching Frequency, Fsw	25 kHz

The following program extracts the small-signal transfer functions of a boost converter with component values as shown in Table 8.5.

```
%This program calculates the small signal transfer function of
the Boost
%converter.

clc

clear all
syms vg rg d rL L rC C R vC iL rds rD vD io

%Converter Dynamical equations
%M1: diL/dt for closed MOSFET.
%M2: dvC/dt for closed MOSFET.
%M3: current of input DC source for closed MOSFET.
%M4: output voltage of converter for closed MOSFET.

%M5: diL/dt for open MOSFET.
%M6: dvC/dt for open MOSFET.
```

```
%M7: current of input DC source for open MOSFET.
%M8: output voltage of converter for open MOSFET.

M1=(-(rg+rL+rds)*iL+vg)/L;
M2=(-vC/(R+rC)+R/(R+rC)*io)/C;
M3=iL;
M4=R/(R+rC)*vC+R*rC/(R+rC)*io;

M5=(-(rg+rL+rD+R*rC/(R+rC))*iL-R/(R+rC)*vC-R*rC/(R+rC)*io
+vg-vD)/L;
M6=((R/(R+rC))*iL-vC/(R+rC)+R/(R+rC)*io)/C;
M7=iL;
M8=R*rC/(R+rC)*iL-R/(R+rC)*vC+R*rC/(R+rC)*io;

%Averaged Equations
diL_dt_ave=simplify(M1*d+M5*(1-d));
dvC_dt_ave=simplify(M2*d+M6*(1-d));
ig_ave=simplify(M3*d+M7*(1-d));
vo_ave=simplify(M4*d+M8*(1-d));

%DC Operating Point
DC=solve(diL_dt_ave==0,dvC_dt_ave==0,'iL','vC');
IL=DC.iL;
VC=DC.vC;

%Linearization
A11=simplify(subs(diff(diL_dt_ave,iL),[iL vC io],[IL VC 0]));
A12=simplify(subs(diff(diL_dt_ave,vC),[iL vC io],[IL VC 0]));
A21=simplify(subs(diff(dvC_dt_ave,iL),[iL vC io],[IL VC 0]));
A22=simplify(subs(diff(dvC_dt_ave,vC),[iL vC io],[IL VC 0]));
AA=[A11 A12;A21 A22];

B11=simplify(subs(diff(diL_dt_ave,io),[iL vC io],[IL VC 0]));
B12=simplify(subs(diff(diL_dt_ave,vg),[iL vC io],[IL VC 0]));
B13=simplify(subs(diff(diL_dt_ave,d),[iL vC io],[IL VC 0]));

B21=simplify(subs(diff(dvC_dt_ave,io),[iL vC io],[IL VC 0]));
B22=simplify(subs(diff(dvC_dt_ave,vg),[iL vC io],[IL VC 0]));
B23=simplify(subs(diff(dvC_dt_ave,d),[iL vC io],[IL VC 0]));

BB=[B11 B12 B13;B21 B22 B23];

C11=simplify(subs(diff(ig_ave,iL),[iL vC io],[IL VC 0]));
C12=simplify(subs(diff(ig_ave,vC),[iL vC io],[IL VC 0]));
```

```
C21=simplify(subs(diff(vo_ave,iL),[iL vC io],[IL VC 0]));
C22=simplify(subs(diff(vo_ave,vC),[iL vC io],[IL VC 0]));
CC=[C11 C12; C21 C22];

D11=simplify(subs(diff(ig_ave,io),[iL vC io],[IL VC 0 ]));
D12=simplify(subs(diff(ig_ave,vg),[iL vC io],[IL VC 0]));
D13=simplify(subs(diff(ig_ave,d),[iL vC io],[IL VC 0]));

D21=simplify(subs(diff(vo_ave,io),[iL vC io],[IL VC 0 ]));
D22=simplify(subs(diff(vo_ave,vg),[iL vC io],[IL VC 0]));
D23=simplify(subs(diff(vo_ave,d),[iL vC io],[IL VC 0]));
DD=[D11 D12 D13;D21 D22 D23];

%Components Values
%Variables have underline are used to store the numeric values of
components
%Variables without underline are symbolic variables.
%for example:
%L: symbolic vvariable shows the inductor inductance
%L_: numeric variable shows the inductor inductance value.
L_=120e-6;
rL_=.01;
C_=100e-6;
rC_=.05;
rds_=.04;
rD_=.01;
VD_=.7;
D_=.6;
VG_=12;
rg_=.1;
R_=50;

AA_=eval(subs(AA,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
BB_=eval(subs(BB,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
CC_=eval(subs(CC,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));
DD_=eval(subs(DD,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0]));

sys=ss(AA_,BB_,CC_,DD_);
sys.stateName={'iL','vC'};
sys.inputname={'io','vg','d'};
sys.outputname={'ig','vo'};

ig_io=sys(1,1);
ig_vg=sys(1,2);
ig_d=sys(1,3);

vo_io=sys(2,1);
vo_vg=sys(2,2);
vo_d=sys(2,3);

Zin=1/ig_vg; %input impedance
Zout=vo_io; %output impedance

%Draws the bode diagram of input/output impedance
figure(1)
bode(Zin), grid minor

figure(2)
bode(Zout), grid minor

%Display the DC operating point of converter
disp('steady state operating point of converter')
disp('IL')
disp(eval(subs(IL,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0])));
disp('VC')
disp(eval(subs(VC,[vg rg rds rD vD rL L rC C R d io],[VG_ rg_ rds_
rD_ VD_ rL_ L_ rC_ C_ R_ D_ 0])));
```

The program gives the following results:

$$\frac{v_o(s)}{d(s)} = -0.007199\frac{\left(s+2\times10^6\right)\left(s-6.703\times10^4\right)}{s^2+1367s+1.356\times10^7}$$

$$Z_{in}(s) = \frac{v_g(s)}{i_g(s)} = 0.00012\frac{s^2+1367s+1.356\times10^7}{s+200} \tag{8.151}$$

$$Z_o(s) = \frac{v_o(s)}{i_o(s)} = 0.049995\frac{\left(s+2\times10^6\right)(s+1160)}{s^2+1367s+1.356\times10^7}$$

Note that the obtained control-to-output transfer function is nonminimum phase like the buck-boost converter. Bode diagram of control-to-output transfer function, open loop input impedance, and open loop output impedance are shown in Figs. 8.84, 8.85, and 8.86, respectively.

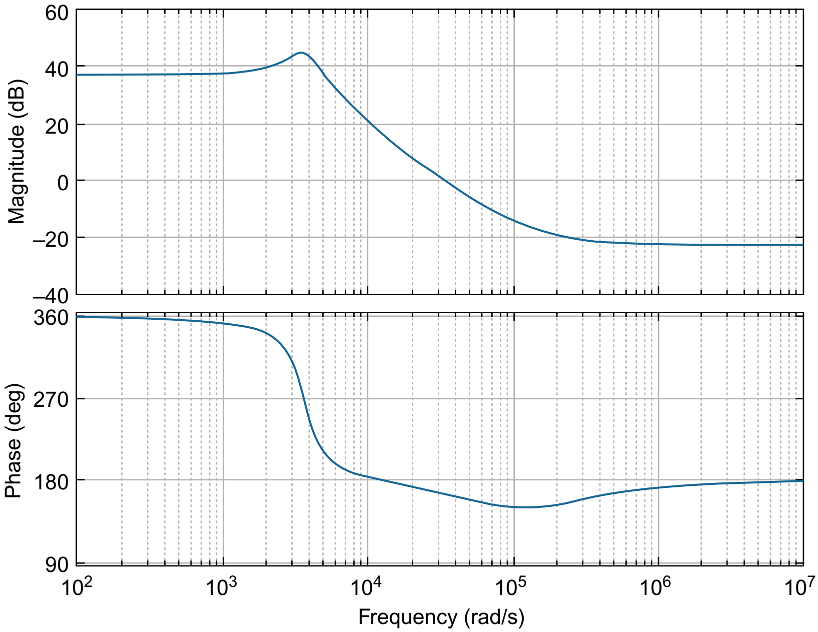


FIG. 8.84 Bode diagram of control-to-output transfer function ($\frac{v_o(s)}{d(s)}$) for the boost converter with parameters as shown in Table 8.5.

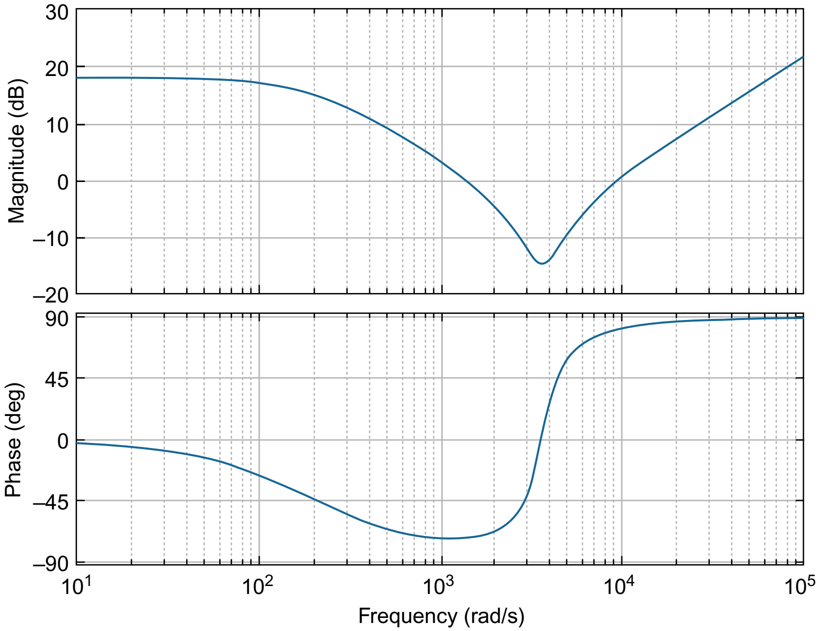


FIG. 8.85 Bode diagram of input impedance ($\frac{v_g(s)}{i_g(s)}$) for the boost converter with parameters as shown in Table 8.5.

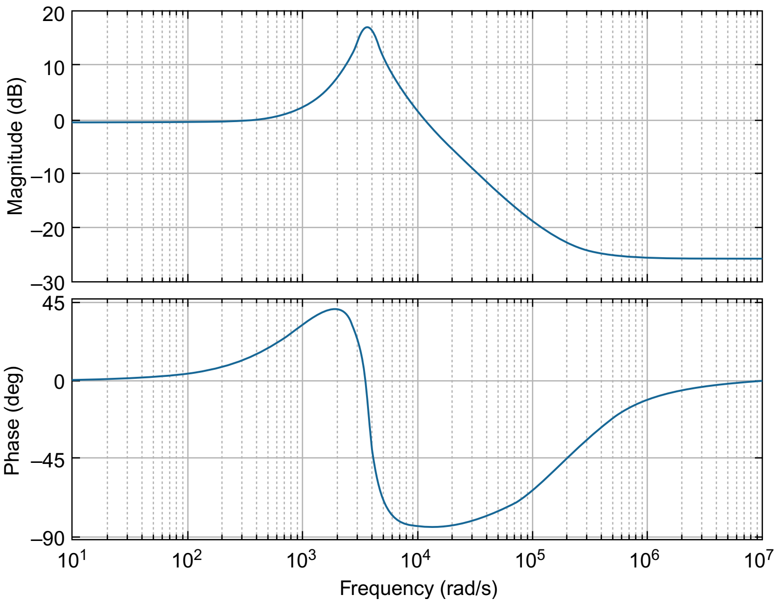


FIG. 8.86 Bode diagram of output impedance ($\frac{\tilde{v}_o(s)}{\tilde{i}_o(s)}$) for the boost converter with parameters as shown in Table 8.5.

8.23 Dynamics of zeta converter

Dynamic model of higher-order converters can be extracted in the same way studied before. In this section, we study the dynamics of a zeta converter, which has fourth-order transfer functions.

Schematic of a zeta converter is shown in Fig. 8.87.

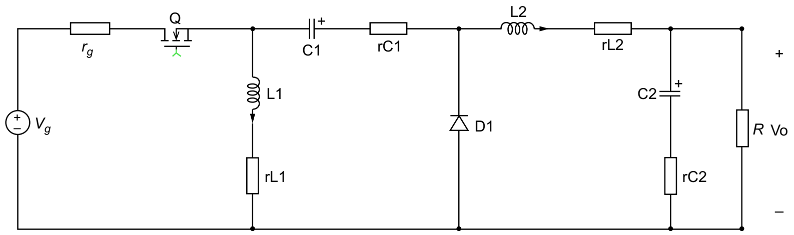


FIG. 8.87 Schematic of zeta converter.

The zeta converter has two switches: a MOSFET switch and a diode. In this schematic, Vg, rg, Li, rLi, Ci, rCi, and R show the input DC source, internal resistance of input DC source, ith inductor, ith inductor Equivalent Series Resistance (ESR), ith capacitor, ith capacitor ESR, and load of converter, respectively. iO is a fictitious current source added to the schematic in order to calculate the output impedance of converter. We assume that converter operates in CCM. The parameters are as shown in Table 8.6.

TABLE 8.6 The zeta converter parameters (see Fig. 8.87).

	Nominal value
Output voltage, Vo	5.2 V
Duty ratio, D	0.23
Input DC source voltage, Vg	20 V
Input DC source internal resistance, rg	0.0 Ω
MOSFET Drain-Source resistance, rds	10 mΩ
Capacitor, C1	100 μF
Capacitor Equivalent Series Resistance(ESR), rC1	0.19 Ω
Capacitor, C2	220 μF
Capacitor Equivalent Series Resistance(ESR), rC2	0.095 Ω
Inductor, L1	100 μH
Inductor ESR, rL1	1 mΩ
Inductor, L2	55 μH
Inductor ESR, rL2	0.55 mΩ
Diode voltage drop, vD	0.7 V
Diode forward resistance, rD	10 mΩ
Load resistor, R	6 Ω
Switching Frequency, Fsw	100 kHz

MOSFET switch is closed for D. T seconds and is open for duration of $(1-D)$. T seconds. D and T show duty ratio and switching period, respectively.

When MOSFET is closed, the diode is reverse biased.

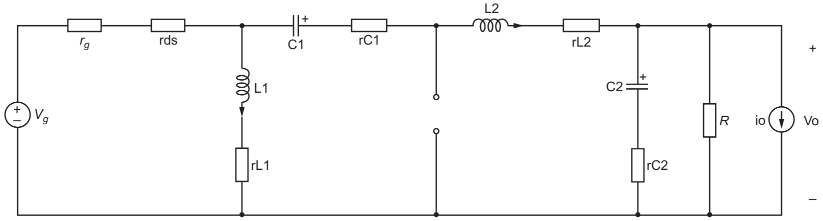


FIG. 8.88 Equivalent circuit of zeta converter when MOSFET is closed.

In this case, the circuit differential equations can be written as:

$$\begin{cases} L_1\dfrac{di_{L_1}}{dt}=-\left(r_{L_1}+r_g+r_{ds}\right)i_{L_1}-\left(r_g+r_{ds}\right)i_{L_2}+v_g \\ L_2\dfrac{di_{L_2}}{dt}=-\left(r_g+r_{ds}\right)i_{L_1}-\left(r_g+r_{ds}+r_{C_1}+r_{L_2}+\dfrac{R\times r_{C_2}}{R+r_{C_2}}\right)i_{L_2}+v_{C_1} \\ \qquad -\dfrac{R}{R+r_{C_2}}v_{C_2}+\dfrac{R\times r_{C_2}}{R+r_{C_2}}i_o+v_g \\ C_1\dfrac{dv_{C_1}}{dt}=-i_{L_2} \\ C_2\dfrac{dv_{C_2}}{dt}=\dfrac{R}{R+r_{C_2}}i_{L_2}-\dfrac{1}{R+r_{C_2}}v_{C_2}-\dfrac{R}{R+r_{C_2}}i_o \end{cases}$$

$$v_o=r_{C_2}C_2\frac{dv_{C_2}}{dt}+v_{C_2}=\frac{R\times r_{C_2}}{R+r_{C_2}}i_{L_2}+\frac{R}{R+r_{C_2}}v_{C_2}-\frac{R\times r_{C_2}}{R+r_{C_2}}i_o \tag{8.152}$$

When MOSFET is opened, the diode is forward biased.

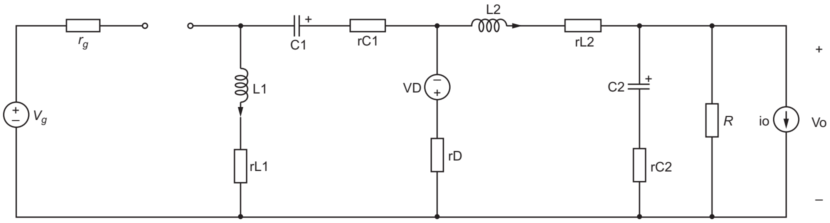


FIG. 8.89 Equivalent circuit of zeta converter when MOSFET is opened. Forward-biased diode is modeled with a voltage source (VD) and a series resistance (rD).

In this case, the circuit differential equations can be written as:

$$\begin{cases} L_1\dfrac{di_{L_1}}{dt}=-(r_{L_1}+r_{C_1}+r_D)i_{L_1}-r_Di_{L_2}-v_{C_1}-v_D \\ L_2\dfrac{di_{L_2}}{dt}=-r_Di_{L_1}-\left(r_D+r_{L_2}+\dfrac{R\times r_{C_2}}{R+r_{C_2}}\right)i_{L_2}-\dfrac{R}{R+r_{C_2}}v_{C_2}+\dfrac{R\times r_{C_2}}{R+r_{C_2}}i_o-v_D \\ C_1\dfrac{dv_{C_1}}{dt}=i_{L_1} \\ C_2\dfrac{dv_{C_2}}{dt}=\dfrac{R}{R+r_{C_2}}i_{L_2}-\dfrac{1}{R+r_{C_2}}v_{C_2}-\dfrac{R}{R+r_{C_2}}i_o \end{cases}$$

$$v_o=r_{C_2}C_2\frac{dv_{C_2}}{dt}+v_{C_2}=\frac{R\times r_{C_2}}{R+r_{C_2}}i_{L_2}+\frac{R}{R+r_{C_2}}v_{C_2}-\frac{R\times r_{C_2}}{R+r_{C_2}}i_o \tag{8.153}$$

Following code extracts the converter transfer functions of a zeta converter with parameters as shown in Table 8.6.

```
% This program calculates the small signal transfer functions of
Zeta converter
clc
clear all

VG=20;       % Average value of input DC source
rg=0;        % Internal resistance of input DC source
rds=.01;     % MOSFET on resistance
C1=100e-6;   % Capacitor C1 value
C2=220e-6;   % Capacitor C2 value
rC1=.19;     % Capacitor C1 Equivalent Series Resistance(ESR)
rC2=.095;    % Capacitor C2 Equivalent Series Resistance(ESR)
L1=100e-6;   % Inductor L1 value
L2=55e-6;    % Inductor L2 value
rL1=1e-3;    % Inductor L1 Equivalent Series Resistance(ESR)
rL2=.55e-3;  % Inductor L2 Equivalent Series Resistance(ESR)
rD=.01;      % Diode series resistance
VD=.7;       % Diode voltage drop
R=6;          % Load resistance
D=.23;       % Duty cylcle
IO=0;        % Average value of output current source
fsw=100e3;   % Switching frequency

syms iL1 iL2 vC1 vC2 io vg vD d
% iL1: Inductor L1 current
% iL2: Inductor L2 current
% vC1: Capacitor C1 voltage
% vC2: Capacitor C2 voltage
% io : Output current source
% vg : Input DC source
% vD : Diode voltage drop
% d  : Duty cycle

%Closed MOSFET Equations
diL1_dt_MOSFET_close=(-(rL1+rg+rds)*iL1-(rg+rds)*iL2+vg)/L1;
diL2_dt_MOSFET_close=(-(rg+rds)*iL1-(rg+rds+rC1+rL2+R*rC2/(R
+rC2))*iL2+vC1-R/(R+rC2)*vC2+R*rC2/(R+rC2)*io+vg)/L2;
dvC1_dt_MOSFET_close=(-iL2)/C1;
dvC2_dt_MOSFET_close=(R/(R+rC2)*iL2-1/(R+rC2)*vC2-R/(R+rC2)
*io)/C2;
vo_MOSFET_close=R*rC2/(R+rC2)*iL2+R/(R+rC2)*vC2-R*rC2/
(R+rC2)*io;

%Opened MOSFET Equations
diL1_dt_MOSFET_open=(-(rL1+rC1+rD)*iL1-rD*iL2-vC1-vD)/L1;
diL2_dt_MOSFET_open=(-rD*iL1-(rD+rL2+R*rC2/(R+rC2))*iL2-R/(R
+rC2)*vC2+R*rC2/(R+rC2)*io-vD)/L2;
```

```
dvC1_dt_MOSFET_open=(iL1)/C1;
dvC2_dt_MOSFET_open=(R/(R+rC2)*iL2-1/(R+rC2)*vC2-R/(R+rC2)
*io)/C2;
vo_MOSFET_open=R*rC2/(R+rC2)*iL2+R/(R+rC2)*vC2-R*rC2/(R+rC2)*io;

%Averaging
averaged_diL1_dt=simplify(d*diL1_dt_MOSFET_close+(1-d)
*diL1_dt_MOSFET_open);
averaged_diL2_dt=simplify(d*diL2_dt_MOSFET_close+(1-d)
*diL2_dt_MOSFET_open);
averaged_dvC1_dt=simplify(d*dvC1_dt_MOSFET_close+(1-d)
*dvC1_dt_MOSFET_open);
averaged_dvC2_dt=simplify(d*dvC2_dt_MOSFET_close+(1-d)
*dvC2_dt_MOSFET_open);
averaged_vo=simplify(d*vo_MOSFET_close+(1-d)*vo_MOSFET_open);

%Substituting the steady values of input DC voltage source, Diode
voltage
%drop, Duty cycle and output current source and calculating the DC
%operating point
right_side_of_averaged_diL1_dt=subs(averaged_diL1_dt,[vg  vD  d
io],[VG VD D IO]);
right_side_of_averaged_diL2_dt=subs(averaged_diL2_dt,[vg  vD  d
io],[VG VD D IO]);
right_side_of_averaged_dvC1_dt=subs(averaged_dvC1_dt,[vg  vD  d
io],[VG VD D IO]);
right_side_of_averaged_dvC2_dt=subs(averaged_dvC2_dt,[vg  vD  d
io],[VG VD D IO]);

DC_OPERATING_POINT=
solve(right_side_of_averaged_diL1_dt==0,right_side_of_avera-
ged_diL2_dt==0,right_side_of_averaged_dvC1_dt==0,right_side_of_
averaged_dvC2_dt==0,'iL1','iL2','vC1','vC2');

IL1=eval(DC_OPERATING_POINT.iL1);
IL2=eval(DC_OPERATING_POINT.iL2);
VC1=eval(DC_OPERATING_POINT.vC1);
VC2=eval(DC_OPERATING_POINT.vC2);
VO=eval(subs(averaged_vo,[iL1 iL2 vC1 vC2 io],[IL1 IL2 VC1
VC2 IO]));

disp('Operating point of converter')
disp('------------------------')
disp('IL1(A)=')
disp(IL1)
disp('IL2(A)=')
disp(IL2)
disp('VC1(V)=')
disp(VC1)
disp('VC2(V)=')
disp(VC2)
disp('VO(V)=')
disp(VO)
disp('---------------------------')

%Linearizing the averaged equations around the DC operating point.
%We want to obtain the matrix A,B,C and D
%     .
%     x=Ax+Bu
%     y=Cx+Du
%
%where,
%     x=[iL1 iL2 vC1 vC2]'
%     u=[io vg d]'
%Since we used the variables D for steady state duty ratio and
C to
%show the capacitors values we use AA, BB, CC and DD instead of A,
%B, C and D.

% Calculating the matrix A
A11=subs(simplify(diff(averaged_diL1_dt,iL1)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A12=subs(simplify(diff(averaged_diL1_dt,iL2)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A13=subs(simplify(diff(averaged_diL1_dt,vC1)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A14=subs(simplify(diff(averaged_diL1_dt,vC2)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);

A21=subs(simplify(diff(averaged_diL2_dt,iL1)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A22=subs(simplify(diff(averaged_diL2_dt,iL2)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A23=subs(simplify(diff(averaged_diL2_dt,vC1)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A24=subs(simplify(diff(averaged_diL2_dt,vC2)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);

A31=subs(simplify(diff(averaged_dvC1_dt,iL1)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
```

```
A32=subs(simplify(diff(averaged_dvC1_dt,iL2)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A33=subs(simplify(diff(averaged_dvC1_dt,vC1)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A34=subs(simplify(diff(averaged_dvC1_dt,vC2)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);

A41=subs(simplify(diff(averaged_dvC2_dt,iL1)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A42=subs(simplify(diff(averaged_dvC2_dt,iL2)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A43=subs(simplify(diff(averaged_dvC2_dt,vC1)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);
A44=subs(simplify(diff(averaged_dvC2_dt,vC2)),[iL1 iL2 vC1 vC2 d
io],[IL1 IL2 VC1 VC2 D IO]);

AA=eval([A11 A12 A13 A14;
         A21 A22 A23 A24;
         A31 A32 A33 A34;
         A41 A42 A43 A44]);

% Calculating the matrix B
B11=subs(simplify(diff(averaged_diL1_dt,io)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
B12=subs(simplify(diff(averaged_diL1_dt,vg)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
B13=subs(simplify(diff(averaged_diL1_dt,d)),[iL1  iL2  vC1  vC2  d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);

B21=subs(simplify(diff(averaged_diL2_dt,io)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
B22=subs(simplify(diff(averaged_diL2_dt,vg)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
B23=subs(simplify(diff(averaged_diL2_dt,d)),[iL1  iL2  vC1  vC2  d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);

B31=subs(simplify(diff(averaged_dvC1_dt,io)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
B32=subs(simplify(diff(averaged_dvC1_dt,vg)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
B33=subs(simplify(diff(averaged_dvC1_dt,d)),[iL1  iL2  vC1  vC2  d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);

B41=subs(simplify(diff(averaged_dvC2_dt,io)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
B42=subs(simplify(diff(averaged_dvC2_dt,vg)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
```

```
B43=subs(simplify(diff(averaged_dvC2_dt,d)),[iL1 iL2 vC1 vC2 d
vD io vg],[IL1 IL2 VC1 VC2 D VD IO VG]);

BB=eval([B11 B12 B13;
         B21 B22 B23;
         B31 B32 B33;
         B41 B42 B43]);

% Calculating the matrix C
C11=subs(simplify(diff(averaged_vo,iL1)),[iL1 iL2 vC1 vC2 d io],
[IL1 IL2 VC1 VC2 D IO]);
C12=subs(simplify(diff(averaged_vo,iL2)),[iL1 iL2 vC1 vC2 d io],
[IL1 IL2 VC1 VC2 D IO]);
C13=subs(simplify(diff(averaged_vo,vC1)),[iL1 iL2 vC1 vC2 d io],
[IL1 IL2 VC1 VC2 D IO]);
C14=subs(simplify(diff(averaged_vo,vC2)),[iL1 iL2 vC1 vC2 d io],
[IL1 IL2 VC1 VC2 D IO]);

CC=eval([C11 C12 C13 C14]);

D11=subs(simplify(diff(averaged_vo,io)),[iL1 iL2 vC1 vC2 d vD io
vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
D12=subs(simplify(diff(averaged_vo,vg)),[iL1 iL2 vC1 vC2 d vD io
vg],[IL1 IL2 VC1 VC2 D VD IO VG]);
D13=subs(simplify(diff(averaged_vo,d)),[iL1 iL2 vC1 vC2 d vD io
vg],[IL1 IL2 VC1 VC2 D VD IO VG]);

% Calculating the matrix D
DD=eval([D11 D12 D13]);

% Producing the State Space Model and obtaining the small signal
transfer
% functions
sys=ss(AA,BB,CC,DD);
sys.inputname={'io';'vg';'d'};
sys.outputname={'vo'};

vo_io=tf(sys(1,1)); %Output impedance transfer function vo(s)/io(s)
vo_vg=tf(sys(1,2)); % vo(s)/vg(s)
vo_d=tf(sys(1,3));  % Control-to-output(vo(s)/d(s))

%drawing the Bode diagrams
figure(1)
bode(vo_io),grid minor,title('vo(s)/io(s)')

figure(2)
bode(vo_vg),grid minor,title('vo(s)/vg(s)')

figure(3)
bode(vo_d),grid minor,title('vo(s)/d(s)')
```

After running the code, following results are obtained:

$$
\begin{aligned}
\frac{\widetilde{v}_o(s)}{\widetilde{i}_o(s)} &= -.093519\frac{\left(s+4.785\times 10^4\right)(s+1163)\left(s^2+1396s+6.882\times 10^7\right)}{\left(s^2+2239s+4.76\times 10^7\right)\left(s^2+2767s+1.026\times 10^8\right)}\\
\frac{\widetilde{v}_o(s)}{\widetilde{v}_g(s)} &= 391.08\frac{\left(s+4.785\times 10^4\right)\left(s^2+1473s+7.7\times 10^7\right)}{\left(s^2+2239s+4.76\times 10^7\right)\left(s^2+2767s+1.026\times 10^8\right)}\\
\frac{\widetilde{v}_o(s)}{\widetilde{d}(s)} &= 43775\frac{\left(s+4.785\times 10^4\right)\left(s^2+1371s+7.696\times 10^7\right)}{\left(s^2+2239s+4.76\times 10^7\right)\left(s^2+2767s+1.026\times 10^8\right)}
\end{aligned}
\tag{8.154}
$$

Bode diagrams of these transfer functions are shown in Figs. 8.90–8.92.

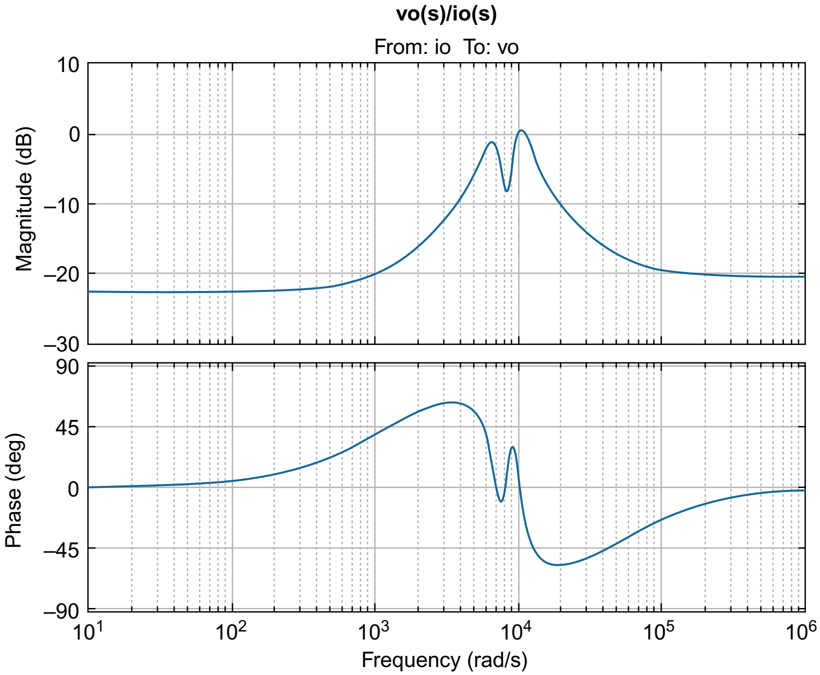


FIG. 8.90 Bode diagram of output impedance ($\frac{\widetilde{v}_o(s)}{\widetilde{i}_o(s)}$) for the zeta converter with parameters as shown in Table 8.6.

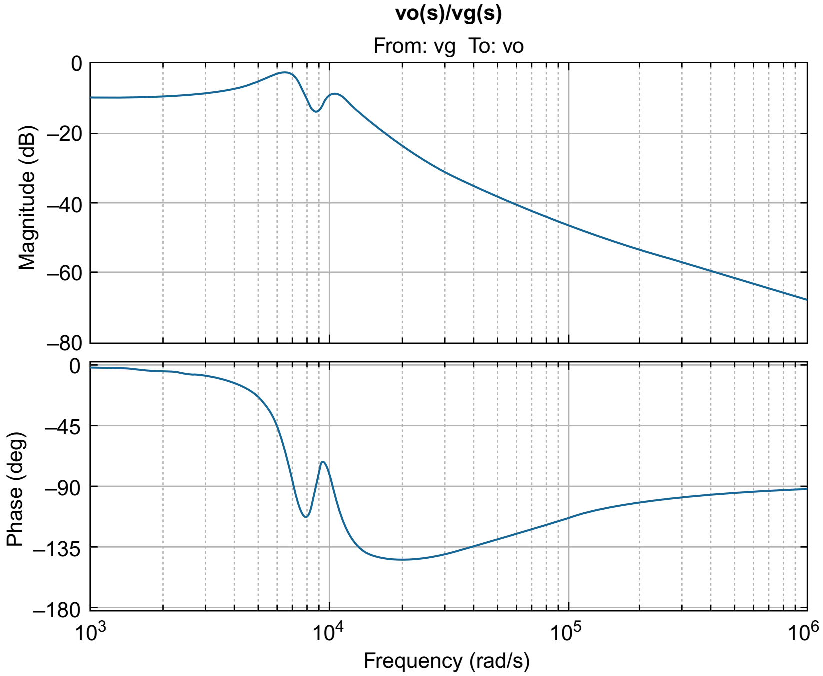


FIG. 8.91 Bode diagram of audio susceptibility ($\frac{\tilde{v}_o(s)}{\tilde{v}_g(s)}$) for the zeta converter with parameters as shown in Table 8.6.

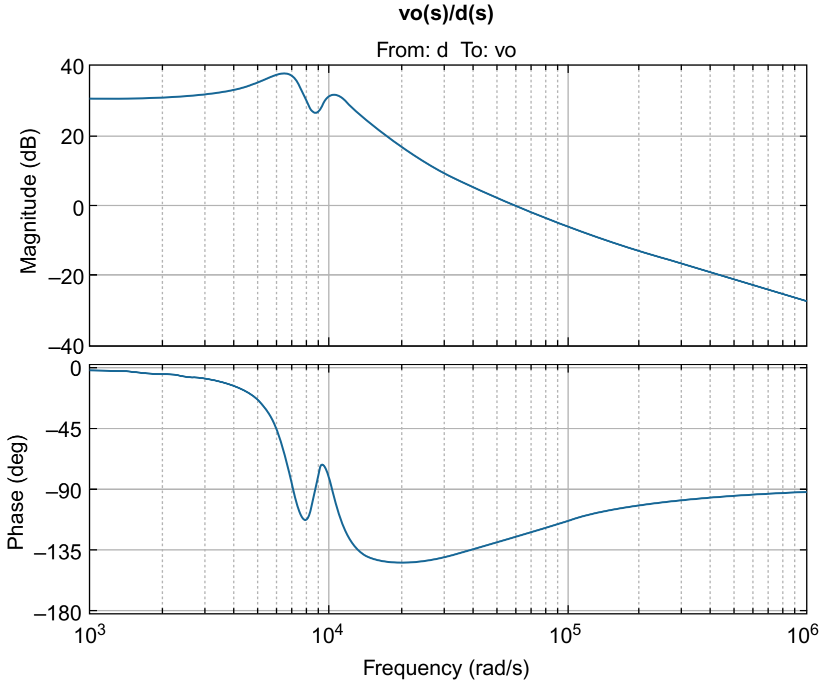


FIG. 8.92 Bode diagram of control-to-output transfer function ($\frac{\tilde{v}_o(s)}{\tilde{d}(s)}$) for the zeta converter with parameters as shown in Table 8.6.

Fig. 8.93 shows the dynamic block diagram of converter.

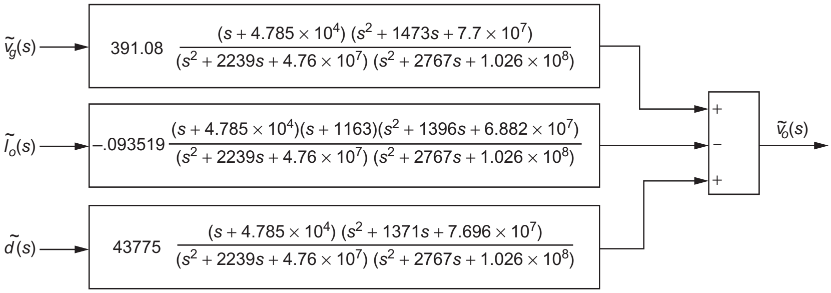

$\tilde{v}_g(s)$
$391.08 \dfrac{(s+4.785\times10^4)(s^2+1473s+7.7\times10^7)}{(s^2+2239s+4.76\times10^7)(s^2+2767s+1.026\times10^8)}$
$\tilde{i}_o(s)$
$-.093519 \dfrac{(s+4.785\times10^4)(s+1163)(s^2+1396s+6.882\times10^7)}{(s^2+2239s+4.76\times10^7)(s^2+2767s+1.026\times10^8)}$
$\tilde{d}(s)$
$43775 \dfrac{(s+4.785\times10^4)(s^2+1371s+7.696\times10^7)}{(s^2+2239s+4.76\times10^7)(s^2+2767s+1.026\times10^8)}$
+
−
+
$\tilde{v}_o(s)$

FIG. 8.93 Dynamical model of zeta converter with parameters given in Table 8.6.

Since the injected test current (io in Figs. 8.88 and 8.89) does not enter the positive end of output voltage, the obtained transfer function for output impedance must be multiplied by -1 in order to be converted to the correct form of output impedance.

PLECS can be used to verify the obtained results. The schematic shown in Fig. 8.94 extracts the output impedance. Extracted output impedance is shown in Fig. 8.95. Obtained result is the same as the one shown in Fig. 8.90 (use cursors to compare different points of two graphs).

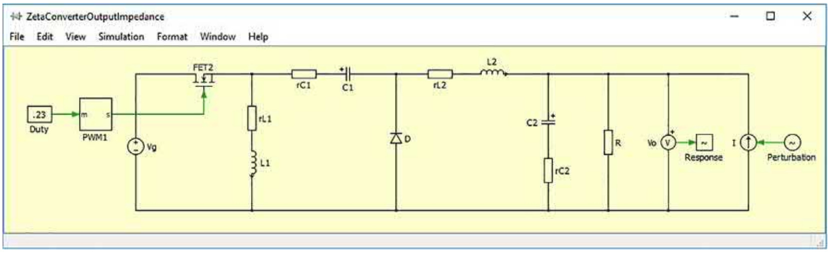

ZetaConverterOutputImpedance
File
Edit
View
Simulation
Format
Window
Help
.23
Duty
m
s
PWM1
Vg
FET2
rL1
L1
rC1
C1
D
rL2
L2
C2
rC2
R
Vo
V
Response
I
Perturbation

FIG. 8.94 Simulation diagram to extract the output impedance ($\frac{\tilde{v}_o(s)}{\tilde{i}_o(s)}$).

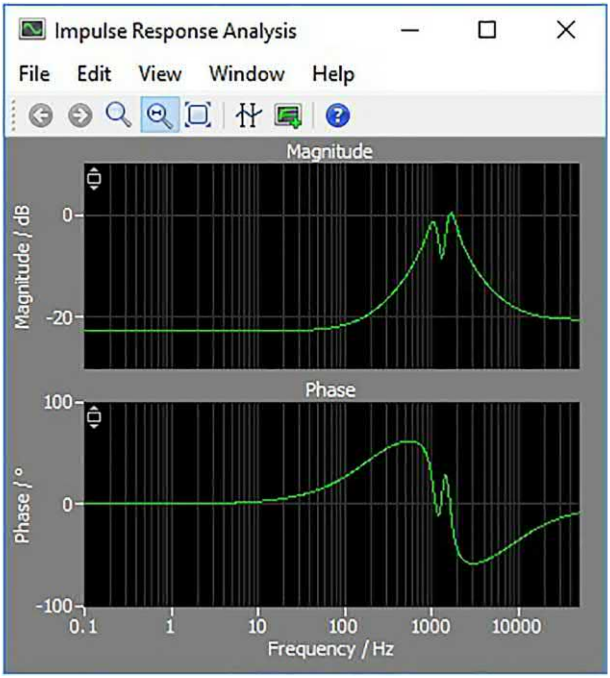


FIG. 8.95 Output impedance of the studied zeta converter (0.1 Hz-50 kHz range). Note that horizontal axis has the unit of Hz. Horizontal axis of Fig. 8.90 has the unit of $\frac{Rad}{s}$.

Schematics to extract the audio susceptibility $(\frac{\tilde{v}_o(s)}{\tilde{v}_g(s)})$ and control-to-output $(\frac{\tilde{v}_o(s)}{\tilde{d}(s)})$ transfer functions are shown in Figs. 8.96 and 8.97, respectively. Analysis results are shown in Figs. 8.98 and 8.99. Obtained results are the same as ones shown in Figs. 8.91 and 8.92.

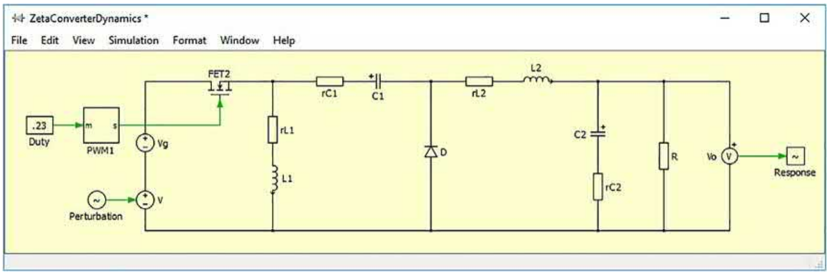

ZetaConverterDynamics *
File
Edit
View
Simulation
Format
Window
Help
.23
Duty
m
s
PWM1
Vg
FET2
rC1
C1
rL2
L2
rL1
L1
D
C2
rC2
R
Vo
V
Response
Perturbation
V

FIG. 8.96 Simulation diagram to extract the audio susceptibility $(\frac{\tilde{v}_o(s)}{\tilde{v}_g(s)})$.

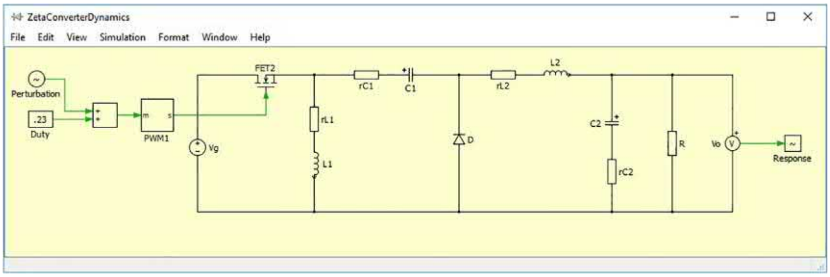

ZetaConverterDynamics
File
Edit
View
Simulation
Format
Window
Help
Perturbation
.23
Duty
m
s
PWM1
Vg
FET2
rL1
L1
rC1
C1
D
rL2
L2
C2
rC2
R
Vo
V
Response

FIG. 8.97 Simulation diagram to extract the control-to-output $(\frac{\tilde{v}_o(s)}{\tilde{d}(s)})$.

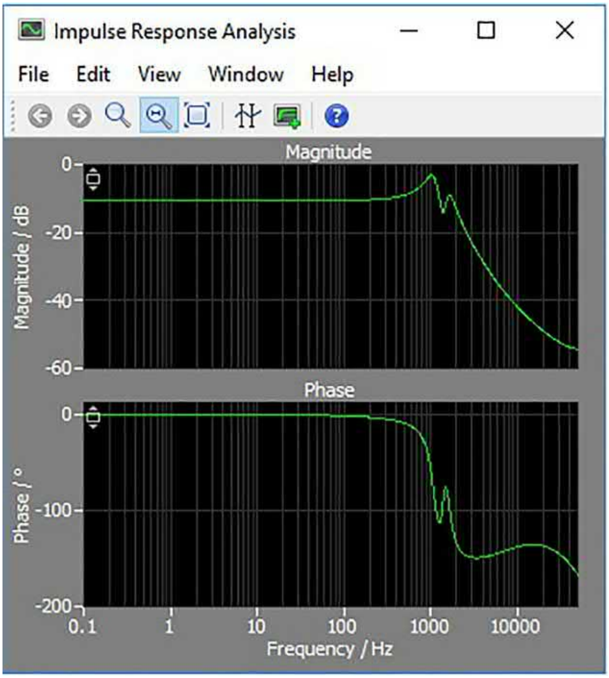


FIG. 8.98 Bode diagram of audio susceptibility $(\frac{\tilde{v}_o(s)}{\tilde{v}_g(s)})$ transfer function for studied zeta converter.

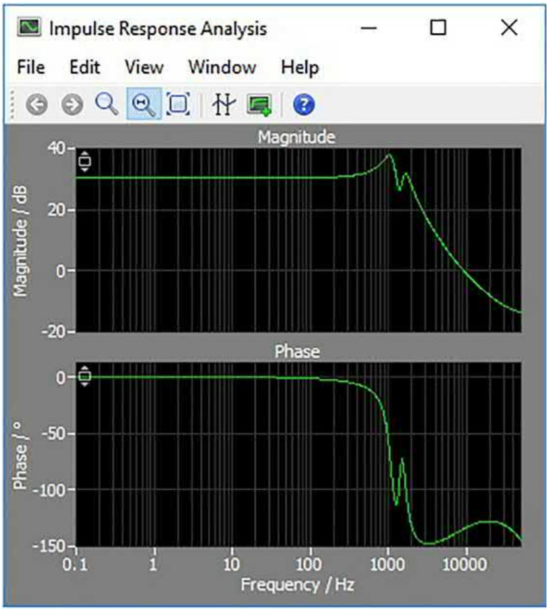

Impulse Response Analysis
File
Edit
View
Window
Help
Magnitude
Magnitude / dB
40
20
0
-20
Phase
Phase / °
0
-50
-100
-150
0.1
1
10
100
1000
10000
Frequency / Hz

FIG. 8.99 Bode diagram of control-to-output $\left(\frac{\tilde{v}_o(s)}{\tilde{d}(s)}\right)$ transfer function for studied zeta converter.

8.24 Inverters

Inverters converts DC to AC. They can be found in applications such as adjustable-speed ac motor drives, uninterruptible power supplies (UPS), and running ac appliances from an automobile battery. The concept of THD (which is studied before) is a good tool to quantify the degree of closeness to a purely sinusoidal waveform. As the THD of inverter decreases, the price of inverter increases.

Fig. 8.100 shows the most basic circuit to convert DC to AC.

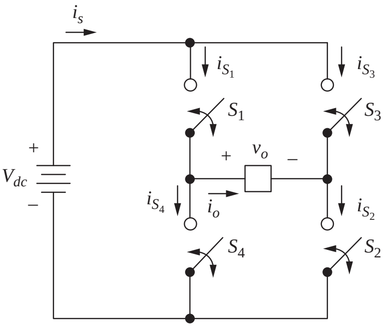

i_s
i_{S_1}
i_{S_3}
S_1
S_3
+
v_o
−
V_{dc}
+
−
i_{S_4}
i_o
i_{S_2}
S_4
S_2

FIG. 8.100 The H bridge inverter.

The circuit is composed of four switches. When the S_1 and S_2 are closed, the load voltage is $+V_{dc}$. When the S_3 and S_4 are closed, the load voltage is $-V_{dc}$. When the S_1 and S_3 (or S_2 and S_4) are closed, the load voltage is 0. S_1 and S_4 should not be closed at the same time (since it produces short circuit), nor should S_2 and S_3.

The simplest way to produce an AC output is closing the S_1 and S_2 for duration of $\frac{T}{2}$ seconds and S_3 and S_4 for the remaining $\frac{T}{2}$ seconds. In this case, the output voltage will be a square wave (Fig. 8.101). Although the output voltage is nonsinusoidal, it may be adequate for some application.

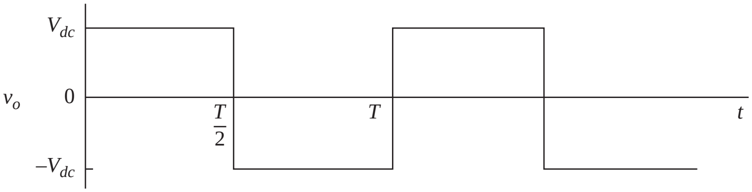


FIG. 8.101 Production of a square wave with the aid of circuit shown in Fig. 8.100. Fourier series of shown waveform is $v_o(t)=\sum_{n\ odd}\frac{4V_{dc}}{n\pi}\sin(n\omega_0 t)$. Amplitude of fundamental harmonic is $\frac{4V_{dc}}{\pi}$. THD of voltage is $THD_V=\frac{\sqrt{V_{rms}{}^2-V_{1,rms}{}^2}}{V_{1,rms}}=\frac{\sqrt{V_{dc}{}^2-\left(\frac{4V_{dc}}{\sqrt{2}\pi}\right)^2}}{\frac{4V_{dc}}{\sqrt{2}\pi}}=0.483$ or 48.3%.

Nature of load current depends on the type of load. For purely resistive loads, the load current will be a square wave. For *RL* type of loads, the current has more of a sinusoidal quality than the voltage because of filtering property of inductance (Fig. 8.102).

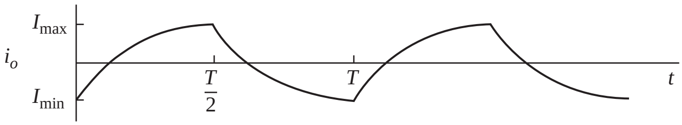


FIG. 8.102 Load current waveform for RL type of loads. Using basic circuit analysis theory, it is quite easy to show that $I_{max}=-I_{min}=\frac{V_{dc}}{R}\left(\frac{1-e^{-\frac{T}{2\tau}}}{1+e^{-\frac{T}{2\tau}}}\right).\tau=\frac{L}{R}$.

When the load is RL, the switches must be bidirectional (Fig. 8.103A). Fig 8.103B shows which pair carries the load current in each instant of time.

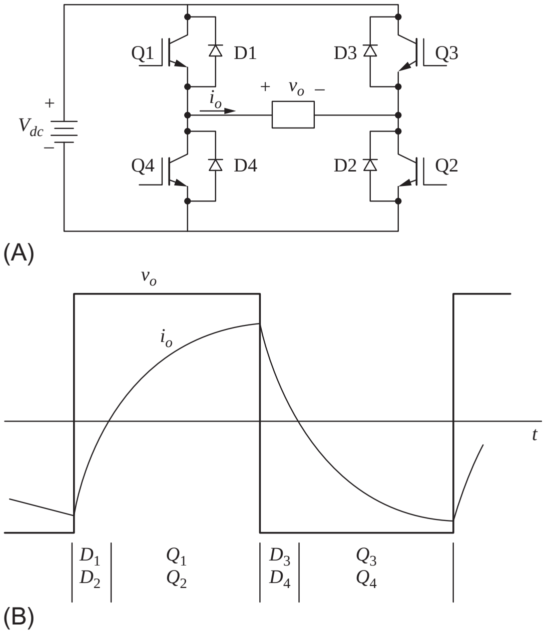


FIG. 8.103 (A) The switches must be bidirectional, i.e., carry current in both directions, when the load is RL. (B) Load current in each instant of time is carried by two switches.

The amplitude of fundamental harmonic of square wave shown in Fig. 8.101 is constant and is determined by the input DC source ($V_1 = \frac{4V_{dc}}{\pi}$). The circuit is capable of producing the 0 V in its output. This can provide a way to control the amplitude of fundamental harmonic of output voltage (Fig. 8.104).

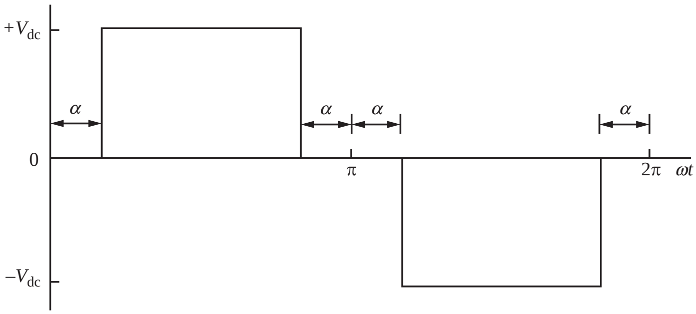


FIG. 8.104 Output voltage can be controlled by adjusting the interval α on each side of the pulse where the output is zero. Fourier series of shown waveforms is $v_o(t) = \sum_{n=1,3,5,\ldots} \left(\frac{4V_{dc}}{n\pi}\right) \cos(n\alpha) \sin(n\omega_0 t)$. RMS of shown waveform is $V_{rms} = V_{dc}\sqrt{1 - \frac{2\alpha}{\pi}}$.

Harmonic content can also be controlled by adjusting α. For instance for $\alpha = 30°$, third harmonic will be zero. In this case, the amplitude of fundamental harmonic will be $V_1 = \frac{4V_{dc}}{\pi}\cos(30°) = 1.1V_{dc}$. nth harmonic can be eliminated by selection of $\alpha = \frac{90}{n}$.

8.24.1 Series H bridge inverters

The THD of output voltage of inverter can be decreased by increasing the number of voltage levels, which the inverter can produce. The H bridge inverters can be connected in series in order to produce more output voltage levels. For instance, Fig. 8.105 shows two H bridge inverters, which are connected in series. Each H bridge inverter produces three different voltage levels ($-V_{dc}$, $0, +V_{dc}$). The output voltage levels of inverter, which is composed of two series H bridge inverters, are $-2V_{dc}, -V_{dc}$, $0, +V_{dc}, +2V_{dc}$. So, the number of output voltage levels increased to five (Fig. 8.106). If we connect five H bridge inverters together, the number of output voltage levels increases to eleven ($-5V_{dc}, -4V_{dc}, -3V_{dc}, -2V_{dc}, -V_{dc}$, $0, +V_{dc}, +2V_{dc}, +3V_{dc}, +4V_{dc}, +5V_{dc}$). Note that each H bridge inverter has its own independent DC source. This is a disadvantage for series of H bridge inverters.

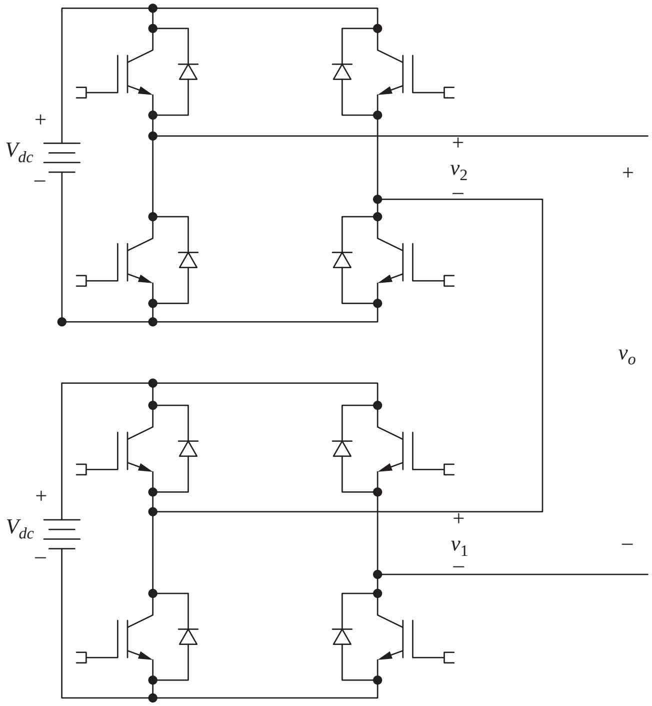


FIG. 8.105 A multilevel inverter composed of series connection of two H bridge inverters.

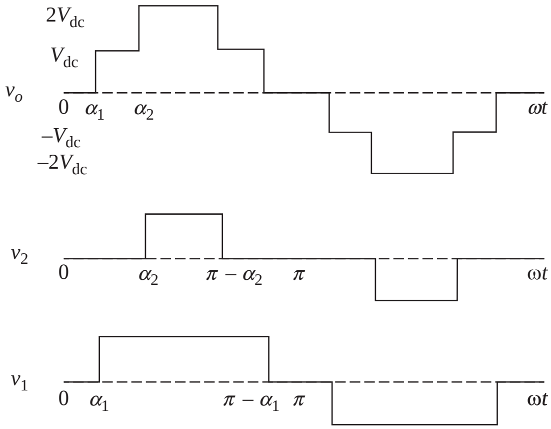


FIG. 8.106 Waveforms of circuit shown in Fig. 8.105. Fourier series of output voltage is $v_o(t)=\frac{4V_{dc}}{\pi}\sum_{n=1,3,5,7,\ldots}[\cos(n\alpha_1)+\cos(n\alpha_2)]\frac{\sin(n\omega_0 t)}{n}$.

8.24.2 Diode-clamped multilevel inverters

Fig. 8.107 shows a diode-clamped multilevel inverter. Note that this circuit uses only one DC source.

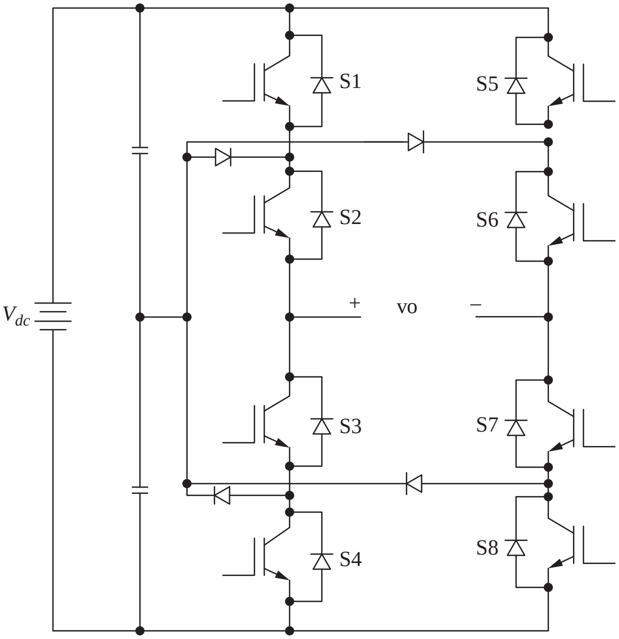


FIG. 8.107 A diode-clamped multilevel inverter implemented with IGBT. The output voltage ($v_o=v_1-v_2$) can have five different levels.

The output voltage of this circuit can have the $-V_{dc}$, $-\frac{V_{dc}}{2}$, 0, $+\frac{V_{dc}}{2}$, and $+V_{dc}$ levels. We will show how these levels are produced. For the analysis, consider only the left half of the bridge, as shown in Fig. 8.108.

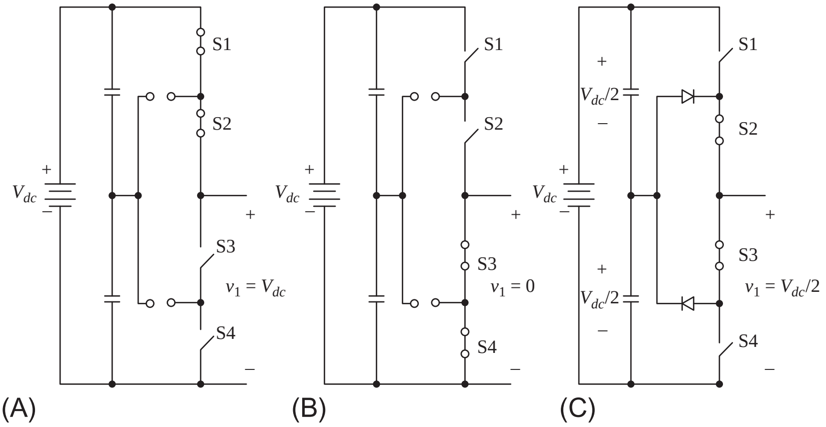


FIG. 8.108 Analysis of one-half of circuit shown in Fig. 8.107 for (A) $v_1 = V_{dc}$, (B) $v_1 = 0$, (C) $v_1 = \frac{V_{dc}}{2}$.

When S_1 and S_2 are closed and S_3 and S_4 are open (Fig. 8.107A), $v_1 = V_{dc}$. The diodes are off for this condition. When S_1 and S_2 are open and S_3 and S_4 are closed (Fig. 8.107B), $v_1 = 0$. The diodes are off in this condition also. When S_1 and S_3 are closed and S_2 and S_4 are open (Fig. 8.107C), $v_1 = \frac{V_{dc}}{2}$.

Using a similar analysis, the right half of the circuit can also produce the voltages V_{dc}, 0, and $\frac{V_{dc}}{2}$. The output voltage is the difference of the voltages between each half-bridge ($v_o = v_1 - v_2$), resulting in the five levels

$$v_o \in \left\{-V_{dc},\ -\frac{V_{dc}}{2},\ 0,\ +\frac{V_{dc}}{2},\ +V_{dc}\right\}$$

More output voltage levels are achieved with additional capacitors and switches. Fig. 8.109 shows the dc source divided across three series capacitors (the voltage across each capacitor is $\frac{V_{dc}}{3}$). The output voltage of this circuit can have seven different levels

$$v_o \in \left\{-V_{dc},\ -\frac{2}{3}V_{dc},\ -\frac{1}{3}V_{dc},\ 0,\ +\frac{1}{3}V_{dc},\ +\frac{2}{3}V_{dc},\ +V_{dc}\right\}$$

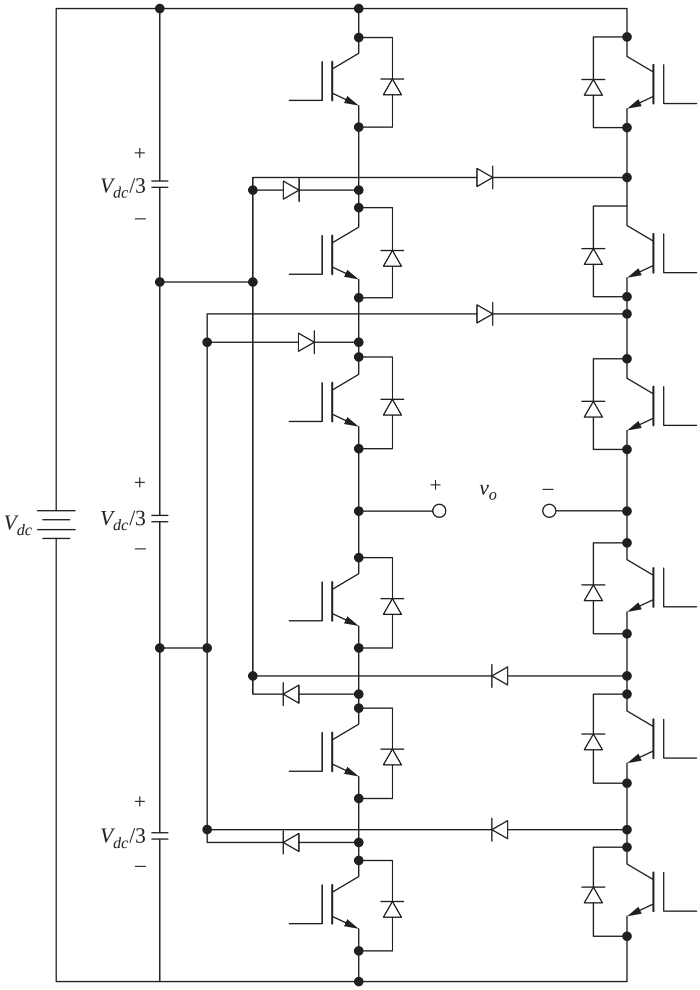


FIG. 8.109 A diode-clamped multilevel inverter, which produces seven output voltage levels.

8.25 Heatsink

Electronic switches dissipate some energy (due to switching and conduction losses) in the form of heat. The produced heat must be removed from the switch in order to keep the internal temperature of device below its maximum.

The produced heat can be removed from the switch more easily with the aid of heatsinks (Figs. 8.110 and 8.111). When the amount of produced heat energy is considerable (for instance, in the inverters, which are used for induction heating), water cooling is used (Fig. 8.112).

FIG. 8.110 A typical heatsink.

FIG. 8.111 A heatsink equipped with a fan.

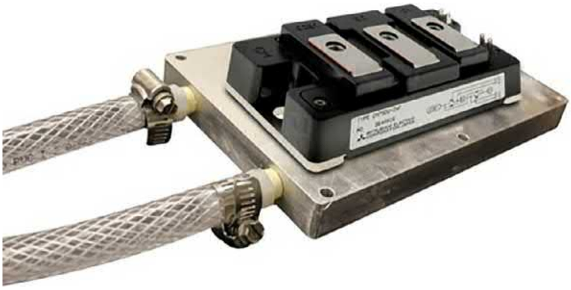


FIG. 8.112 When the amount of produced heat in the switch is considerable, water cooling is used.

The heat transfer problem is quite a complex problem. Exact solution of heat transfer problem requires finite element softwares. However, for the purpose of power electronics converters, there are simplified estimations, which can be used to get rid of complex heat transfer mathematics.

The heat can be transferred in three ways:

1. Conduction
2. Convection
3. Radiation

We assume that produced heat in the switch is transferred to the ambient using the conduction only. This is simplifying assumption and in real world, the produced heat is transferred using convection and radiations as well.

In general, the temperature difference between two points is a function of thermal power and thermal resistance. Thermal resistance is defined as

$$R_\theta = \frac{T_1 - T_2}{P} \tag{8.155}$$

where R_θ = thermal resistance (°C/W), $T_1 - T_2$= temperature difference (°C), and P= thermal power (W).

Fig. 8.113 shows the electric circuit equivalent of the equation. The thermal power is modeled as a current source, thermal resistance is modeled as a electrical resistance, and temperature difference is modeled as the voltage difference.

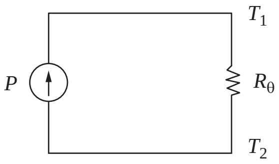


FIG. 8.113 An electric circuit equivalent of Eq. (8.155).

The internal temperature of an electronic switching device is referred to as the junction temperature. Although devices such as MOSFETs do not have a junction per se when conducting, the term is still used. In an electronic device without a heat sink, the junction temperature is determined by thermal power and the junction-to-ambient thermal resistance $R_{\theta,\,JA}$. The ambient temperature is that of the air in contact with the case. The $R_{\theta,\,JA}$ can be found in the datasheet of MOSFET.

For instance, assume that $R_{\theta,JA} = 62\,°\mathrm{C/W}$, ambient temperature is 40 °C, and the maximum junction temperature ($T_{J,max}$) is 175 °C. The maximum power that MOSFET can absorb is 2.177 W since (see Fig. 8.114).

$$P = \frac{T_1 - T_2}{R_\theta} = \frac{T_J - T_A}{R_{\theta,JA}} = \frac{175 - 40}{62} = 2.177\,W \tag{8.156}$$

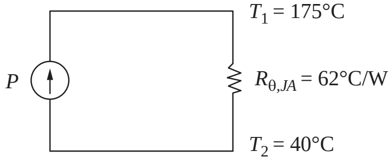


FIG. 8.114 Electric circuit equivalent of heat transfer in a MOSFET without heatsink. The temperature of junction is allowed to reach its maximum.

Assume that we wish the junction temperature not to increase above 150 ° C for increased reliablity. In this case, the maximum power that MOSFET can absorb decreases to 1.77 W since (see Fig. 8.115)

$$P = \frac{T_1 - T_2}{R_\theta} = \frac{T_J - T_A}{R_{\theta,JA}} = \frac{150 - 40}{62} = 1.77\,W \tag{8.157}$$

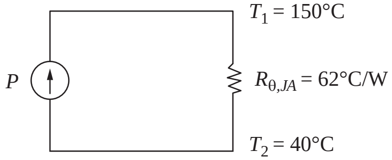


FIG. 8.115 Electric circuit equivalent of heat transfer in a MOSFET without heatsink. The temperature of junction is limited to 150 ° C.

So, a device without any heatsink can dissipate a little power. Generally, a heatsink is required to dissipate the produced heat safely. A heatsink reduces the thermal resistance from the junction to ambient. The heatsinks are produced in all sizes. Generally, the heatsinks are produced from aluminum (Fig. 8.116).

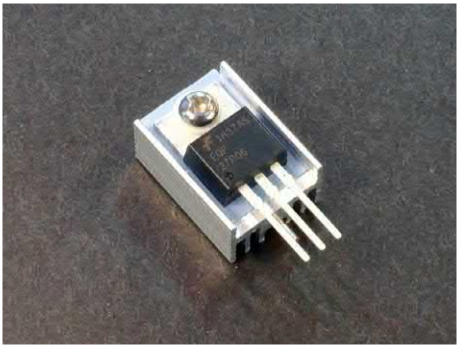


FIG. 8.116 A MOSFET switch with heatsink. The case of the device is often attached to the heat sink with a thermal compound to fill the small voids between the imperfect surfaces of the case and sink.

When the switch is mounted on heatsink, the thermal power flows from the junction to the case, from the case to the heat sink, and then from the heat sink to ambient. The corresponding thermal resistances are $R_{\theta,JC}$, $R_{\theta,CS}$, and $R_{\theta,SA}$ as shown in Fig. 8.117. Value of $R_{\theta,JC}$, $R_{\theta,CS}$ can be found in the switch (MOSFET, IGBT, …) datasheet. $R_{\theta,SA}$ can be found in the heatsink datasheet.

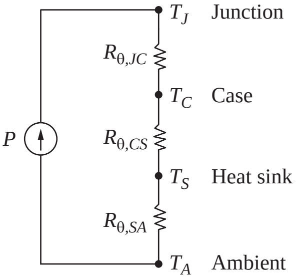

T_J Junction
$R_{\theta,JC}$
T_C Case
P
$R_{\theta,CS}$
T_S Heat sink
$R_{\theta,SA}$
T_A Ambient

FIG. 8.117 The electrical-equivalent circuit for a switch mounted on a heatsink.

The voltage of different nodes of Fig. 8.117 (which represents temperatures) can be calculated easily:

$$\begin{aligned}
&T_S = P \times R_{\theta,SA} + T_A \\
&T_C = P \times R_{\theta,CS} + T_S = P \times (R_{\theta,CS} + R_{\theta,SA}) + T_A \\
&T_J = P \times R_{\theta,JC} + T_C = P \times (R_{\theta,JC} + R_{\theta,CS} + R_{\theta,SA}) + T_A
\end{aligned} \tag{8.158}$$

For instance, assume that $R_{\theta,JC} = 1.87\,°\text{C/W}$, $R_{\theta,CS} = 0.5\,°\text{C/W}$, $R_{\theta,SA} = 7.2\,°\text{C/W}$, $T_{Ambient} = 40\,°\text{C}$, and $T_{J,max} = 150\,°\text{C}$. The equivalent electric circuit shown in Fig. 8.118 can be drawn for this case.

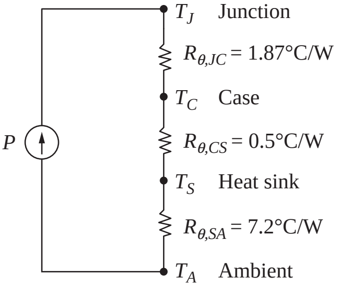

T_J Junction
$R_{\theta,JC}$ = 1.87°C/W
T_C Case
P
$R_{\theta,CS}$ = 0.5°C/W
T_S Heat sink
$R_{\theta,SA}$ = 7.2°C/W
T_A Ambient

FIG. 8.118 Equivalent electric circuit for the studied example.

The maximum power, which can be absorbed by the switch is:

$$P = \frac{T_J - T_A}{R_{\theta,JC} + R_{\theta,CS} + R_{\theta,SA}} = \frac{150 - 40}{1.87 + 0.5 + 7.2} = \frac{110}{9.57} = 11.5\,W \tag{8.159}$$

So, with the aid of a heatsink with thermal resistance of $R_{\theta,SA} = 7.2\,^\circ\mathrm{C/W}$, maximum of 11.5 W is dissipatable. Assume that the power to be dissipated is 15 W. In this case, the junction temperature is

$$T_J = P(R_{\theta,JC} + R_{\theta,CS} + R_{\theta,SA}) + T_A = 15(1.87 + 0.5 + 7.2) + 40 = 184°C \tag{8.160}$$

So, the switch will burn out since the junction temperature is above the maximum allowed temperature. In order to dissipate 15 W safely, a heatsink with thermal resistance of $R_{\theta,SA} = 4.96\,^\circ\mathrm{C/W}$ is required:

$$R_{\theta,SA} = \frac{T_J - T_A}{P} - R_{\theta,JC} - R_{\theta,CS} = \frac{150 - 40}{15} - 1.87 - 0.5 = 4.96°\mathrm{C/W} \tag{8.161}$$

Further reading

[1] D. Hart, Power Electronics, Mc Graw Hill, 2011.

[2] N. Mohan, T. Undeland, Power Electronics Converters, Applications and Design, Wiley, 2002.

[3] M. Rashid, Power Electronics Devices, Circuits and Applications, Pearson, 2013.

[4] R. Erikson, D. Maksimovic, Fundamentals of Power Electronics, Springer, 2001.

[5] S. Ang, A. Oliva, Power Switching Converters, Taylor & Francis, 2005.

[6] Marian K. Kazimierczuck, Pulse Width Modulated DC-DC Power Converters, John Wiley, 2012.

[7] F.L. Luo, H. Ye, Advanced DC/AC Inverters: Applications in Renewable Energy, CRC Press, 2013.

[8] J. Sun, D.M. Mitchel, et al., Averaged modeling of PWM converters operating in discontinuous conduction mode, IEEE Trans. Power Elect. 16(4) (2001) 482–492.

[9] F. Asadi, N. Abut, et al., A simple method for electrical machine's mechanical parameter extraction, Int. J. Adv. Appl. Sci. 4(4) (2017) 164–169.

[10] F. Asadi, N. Abut, et al., A novel buck-boost converter, Int. J. Adv. Appl. Sci. 4(5) (2017) 133–137.

[11] F. Asadi, N. Abut, et al., Linear controller design for a large dc gain converter, Int. J. Adv. Appl. Sci. 4(7) (2017) 136–140.

[12] F. Asadi, N. Abut, et al., Dynamics and control of a novel buck-boost converter with low stresses on switches and diodes, Int. J. Adv. Appl. Sci. 4(8) (2017) 149–153.

[13] W. Do, F. Asadi, et al., A new analysis way of three-phase switched capacitor converters, J. Circ. Syst. Comput. 28(08) (2019) 1–19.

Appendix A

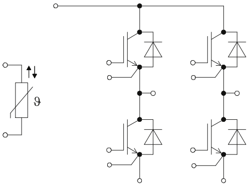
Technische Information / technical information

IGBT-Module
IGBT-modules

F4-50R06W1E3

EasyPACK Modul mit Trench/Feldstopp IGBT3 und Emitter Controlled 3 Diode und NTC
EasyPACK module with Trench/Fieldstop IGBT3 and Emitter Controlled 3 diode and NTC

Typical Appearance



V_{CES} = 600V
$I_{C\ nom}$ = 50A / I_{CRM} = 100A

Typische Anwendungen
- Hilfsumrichter
- Induktives Erwärmen und Schweißen
- Servoumrichter
- USV-Systeme

Typical Applications
- Auxiliary Inverters
- Inductive Heating and Welding
- Servo Drives
- UPS Systems

Elektrische Eigenschaften
- Niederinduktives Design
- Niedrige Schaltverluste
- Trench IGBT 3
- niedriges V_{CEsat}

Electrical Features
- Low inductive design
- Low Switching Losses
- Trench IGBT 3
- Low V_{CEsat}

Mechanische Eigenschaften
- Al_2O_3 Substrat für kleinen thermischen Widerstand
- Kompaktes Design
- Lötverbindungs Technologie
- Robuste Montage durch integrierte Befestigungsklammern

Mechanical Features
- Al_2O_3 Substrate for Low Thermal Resistance
- Compact Design
- Solder Contact Technology
- Rugged mounting due to integrated mounting clamps

Module Label Code

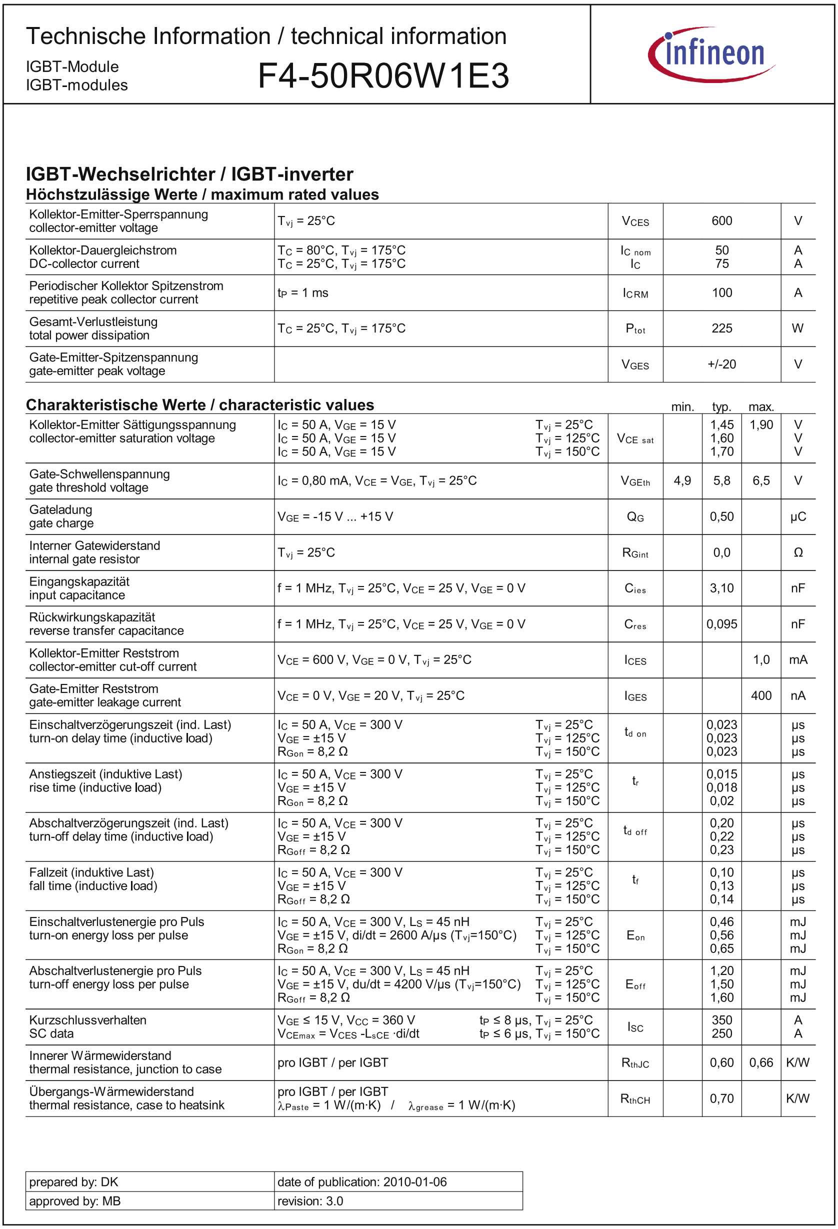
Barcode Code 128

00000123456000000000000

DMX - Code

Content of the Code	**Digit**
Module Serial Number	1 - 5
Module Material Number	6 - 11
Production Order Number	12 - 19
Datecode (Production Year)	20 - 21
Datecode (Production Week)	22 - 23

prepared by: DK	date of publication: 2010-01-06	material no: 28311
approved by: MB	revision: 3.0	UL approved (E83335)



Technische Information / technical information

IGBT-Module
IGBT-modules

F4-50R06W1E3

Infineon

IGBT-Wechselrichter / IGBT-inverter

Höchstzulässige Werte / maximum rated values

Kollektor-Emitter-Sperrspannung collector-emitter voltage	T_{vj} = 25°C	V_{CES}	600	V
Kollektor-Dauergleichstrom DC-collector current	T_C = 80°C, T_{vj} = 175°C T_C = 25°C, T_{vj} = 175°C	$I_{C\ nom}$ I_C	50 75	A A
Periodischer Kollektor Spitzenstrom repetitive peak collector current	t_P = 1 ms	I_{CRM}	100	A
Gesamt-Verlustleistung total power dissipation	T_C = 25°C, T_{vj} = 175°C	P_{tot}	225	W
Gate-Emitter-Spitzenspannung gate-emitter peak voltage		V_{GES}	+/-20	V

Charakteristische Werte / characteristic values

				min.	typ.	max.	
Kollektor-Emitter Sättigungsspannung collector-emitter saturation voltage	I_C = 50 A, V_{GE} = 15 V I_C = 50 A, V_{GE} = 15 V I_C = 50 A, V_{GE} = 15 V	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	$V_{CE\ sat}$		1,45 1,60 1,70	1,90	V V V
Gate-Schwellenspannung gate threshold voltage	I_C = 0,80 mA, V_{CE} = V_{GE}, T_{vj} = 25°C		V_{GEth}	4,9	5,8	6,5	V
Gateladung gate charge	V_{GE} = -15 V ... +15 V		Q_G		0,50		µC
Interner Gatewiderstand internal gate resistor	T_{vj} = 25°C		R_{Gint}		0,0		Ω
Eingangskapazität input capacitance	f = 1 MHz, T_{vj} = 25°C, V_{CE} = 25 V, V_{GE} = 0 V		C_{ies}		3,10		nF
Rückwirkungskapazität reverse transfer capacitance	f = 1 MHz, T_{vj} = 25°C, V_{CE} = 25 V, V_{GE} = 0 V		C_{res}		0,095		nF
Kollektor-Emitter Reststrom collector-emitter cut-off current	V_{CE} = 600 V, V_{GE} = 0 V, T_{vj} = 25°C		I_{CES}			1,0	mA
Gate-Emitter Reststrom gate-emitter leakage current	V_{CE} = 0 V, V_{GE} = 20 V, T_{vj} = 25°C		I_{GES}			400	nA
Einschaltverzögerungszeit (ind. Last) turn-on delay time (inductive load)	I_C = 50 A, V_{CE} = 300 V V_{GE} = ±15 V R_{Gon} = 8,2 Ω	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	$t_{d\ on}$		0,023 0,023 0,023		µs µs µs
Anstiegszeit (induktive Last) rise time (inductive load)	I_C = 50 A, V_{CE} = 300 V V_{GE} = ±15 V R_{Gon} = 8,2 Ω	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	t_r		0,015 0,018 0,02		µs µs µs
Abschaltverzögerungszeit (ind. Last) turn-off delay time (inductive load)	I_C = 50 A, V_{CE} = 300 V V_{GE} = ±15 V R_{Goff} = 8,2 Ω	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	$t_{d\ off}$		0,20 0,22 0,23		µs µs µs
Fallzeit (induktive Last) fall time (inductive load)	I_C = 50 A, V_{CE} = 300 V V_{GE} = ±15 V R_{Goff} = 8,2 Ω	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	t_f		0,10 0,13 0,14		µs µs µs
Einschaltverlustenergie pro Puls turn-on energy loss per pulse	I_C = 50 A, V_{CE} = 300 V, L_S = 45 nH V_{GE} = ±15 V, di/dt = 2600 A/µs (T_{vj}=150°C) R_{Gon} = 8,2 Ω	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	E_{on}		0,46 0,56 0,65		mJ mJ mJ
Abschaltverlustenergie pro Puls turn-off energy loss per pulse	I_C = 50 A, V_{CE} = 300 V, L_S = 45 nH V_{GE} = ±15 V, du/dt = 4200 V/µs (T_{vj}=150°C) R_{Goff} = 8,2 Ω	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	E_{off}		1,20 1,50 1,60		mJ mJ mJ
Kurzschlussverhalten SC data	V_{GE} ≤ 15 V, V_{CC} = 360 V V_{CEmax} = V_{CES} -L_{sCE} ·di/dt	t_P ≤ 8 µs, T_{vj} = 25°C t_P ≤ 6 µs, T_{vj} = 150°C	I_{SC}		350 250		A A
Innerer Wärmewiderstand thermal resistance, junction to case	pro IGBT / per IGBT		R_{thJC}		0,60	0,66	K/W
Übergangs-Wärmewiderstand thermal resistance, case to heatsink	pro IGBT / per IGBT λ_{Paste} = 1 W/(m·K) / λ_{grease} = 1 W/(m·K)		R_{thCH}		0,70		K/W

prepared by: DK	date of publication: 2010-01-06
approved by: MB	revision: 3.0

Technische Information / technical information

IGBT-Module
IGBT-modules

F4-50R06W1E3

Diode-Wechselrichter / diode-inverter

Höchstzulässige Werte / maximum rated values

Periodische Spitzensperrspannung repetitive peak reverse voltage	T_{vj} = 25°C	V_{RRM}	600	V
Dauergleichstrom DC forward current		I_F	50	A
Periodischer Spitzenstrom repetitive peak forward current	t_P = 1 ms	I_{FRM}	100	A
Grenzlastintegral I²t - value	V_R = 0 V, t_P = 10 ms, T_{vj} = 125°C V_R = 0 V, t_P = 10 ms, T_{vj} = 150°C	I^2t	370 330	A²s A²s

Charakteristische Werte / characteristic values

				min.	typ.	max.	
Durchlassspannung forward voltage	I_F = 50 A, V_{GE} = 0 V I_F = 50 A, V_{GE} = 0 V I_F = 50 A, V_{GE} = 0 V	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	V_F		1,55 1,50 1,45	1,95	V V V
Rückstromspitze peak reverse recovery current	I_F = 50 A, - dI_F/dt = 2600 A/µs (T_{vj}=150°C) V_R = 300 V V_{GE} = -15 V	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	I_{RM}		60,0 68,0 72,0		A A A
Sperrverzögerungsladung recovered charge	I_F = 50 A, - dI_F/dt = 2600 A/µs (T_{vj}=150°C) V_R = 300 V V_{GE} = -15 V	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	Q_r		2,10 3,40 3,95		µC µC µC
Abschaltenergie pro Puls reverse recovery energy	I_F = 50 A, - dI_F/dt = 2600 A/µs (T_{vj}=150°C) V_R = 300 V V_{GE} = -15 V	T_{vj} = 25°C T_{vj} = 125°C T_{vj} = 150°C	E_{rec}		0,42 0,71 0,83		mJ mJ mJ
Innerer Wärmewiderstand thermal resistance, junction to case	pro Diode / per diode		R_{thJC}		0,80	0,90	K/W
Übergangs-Wärmewiderstand thermal resistance, case to heatsink	pro Diode / per diode λ_{Paste} = 1 W/(m·K) / λ_{grease} = 1 W/(m·K)		R_{thCH}		0,70		K/W

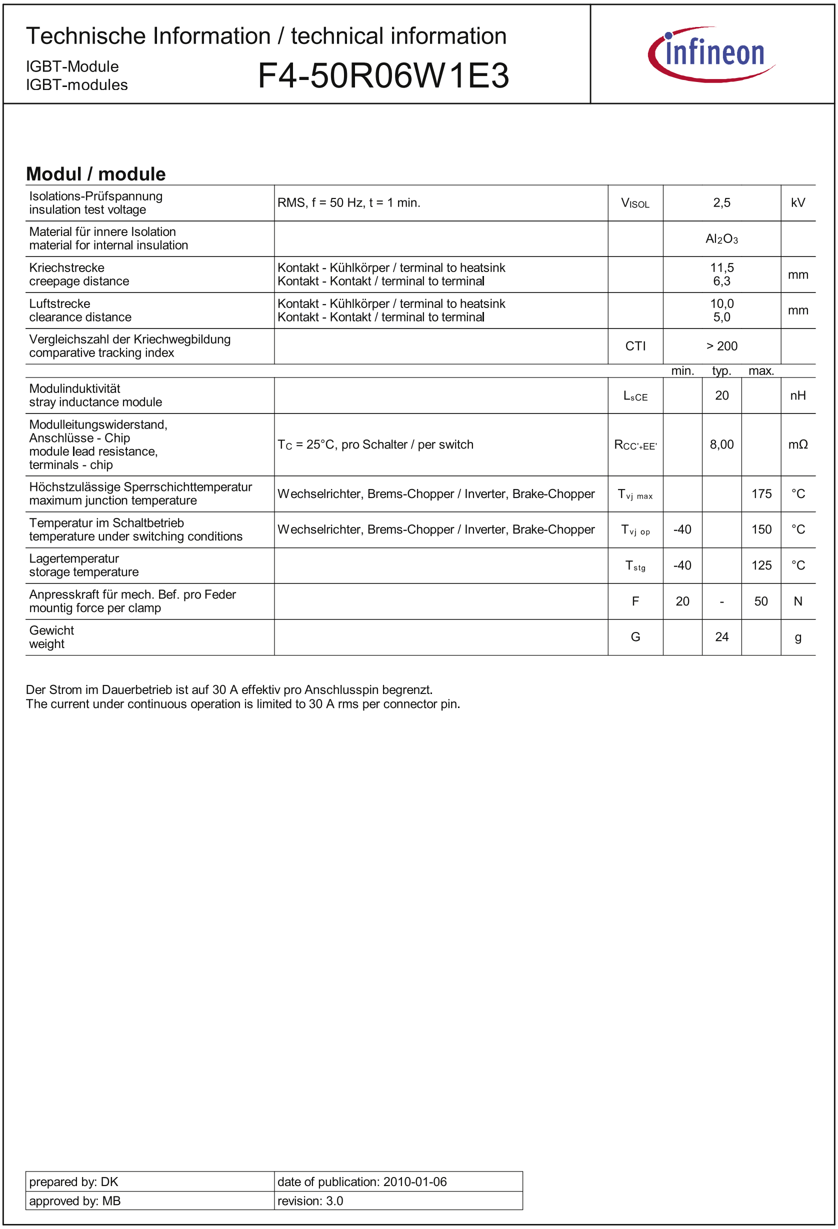
NTC-Widerstand / NTC-thermistor

Charakteristische Werte / characteristic values

			min.	typ.	max.	
Nennwiderstand rated resistance	T_C = 25°C	R_{25}		5,00		kΩ
Abweichung von R_{100} deviation of R_{100}	T_C = 100°C, R_{100} = 493 Ω	ΔR/R	-5		5	%
Verlustleistung power dissipation	T_C = 25°C	P_{25}			20,0	mW
B-Wert B-value	$R_2 = R_{25} \exp [B_{25/50}(1/T_2 - 1/(298,15\ K))]$	$B_{25/50}$		3375		K
B-Wert B-value	$R_2 = R_{25} \exp [B_{25/80}(1/T_2 - 1/(298,15\ K))]$	$B_{25/80}$		3411		K
B-Wert B-value	$R_2 = R_{25} \exp [B_{25/100}(1/T_2 - 1/(298,15\ K))]$	$B_{25/100}$		3433		K

Angaben gemäß gültiger Application Note.
Specification according to the valid application note.

prepared by: DK	date of publication: 2010-01-06
approved by: MB	revision: 3.0



Technische Information / technical information

IGBT-Module
IGBT-modules

F4-50R06W1E3

Modul / module

			min.	typ.	max.	
Isolations-Prüfspannung insulation test voltage	RMS, f = 50 Hz, t = 1 min.	V_{ISOL}		2,5		kV
Material für innere Isolation material for internal insulation				Al_2O_3		
Kriechstrecke creepage distance	Kontakt - Kühlkörper / terminal to heatsink Kontakt - Kontakt / terminal to terminal			11,5 6,3		mm
Luftstrecke clearance distance	Kontakt - Kühlkörper / terminal to heatsink Kontakt - Kontakt / terminal to terminal			10,0 5,0		mm
Vergleichszahl der Kriechwegbildung comparative tracking index		CTI		> 200		
Modulinduktivität stray inductance module		L_{sCE}		20		nH
Modulleitungswiderstand, Anschlüsse - Chip module lead resistance, terminals - chip	T_C = 25°C, pro Schalter / per switch	$R_{CC'+EE'}$		8,00		mΩ
Höchstzulässige Sperrschichttemperatur maximum junction temperature	Wechselrichter, Brems-Chopper / Inverter, Brake-Chopper	$T_{vj\,max}$			175	°C
Temperatur im Schaltbetrieb temperature under switching conditions	Wechselrichter, Brems-Chopper / Inverter, Brake-Chopper	$T_{vj\,op}$	-40		150	°C
Lagertemperatur storage temperature		T_{stg}	-40		125	°C
Anpresskraft für mech. Bef. pro Feder mountig force per clamp		F	20	-	50	N
Gewicht weight		G		24		g

Der Strom im Dauerbetrieb ist auf 30 A effektiv pro Anschlusspin begrenzt.
The current under continuous operation is limited to 30 A rms per connector pin.

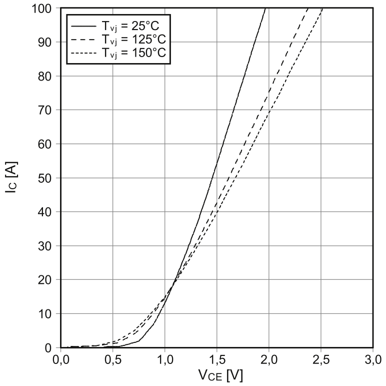
prepared by: DK	date of publication: 2010-01-06
approved by: MB	revision: 3.0

Technische Information / technical information

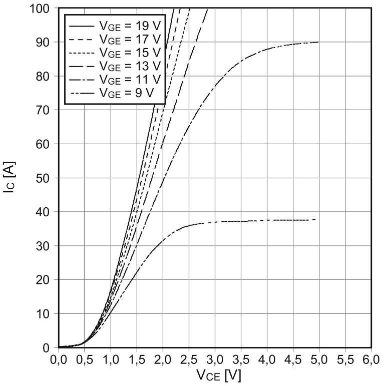
IGBT-Module
IGBT-modules

F4-50R06W1E3

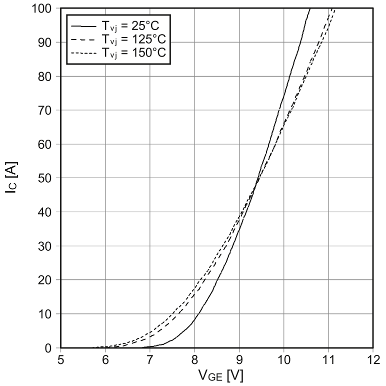
Ausgangskennlinie IGBT-Wechselr. (typisch)
output characteristic IGBT-inverter (typical)
$I_C = f(V_{CE})$
$V_{GE} = 15$ V



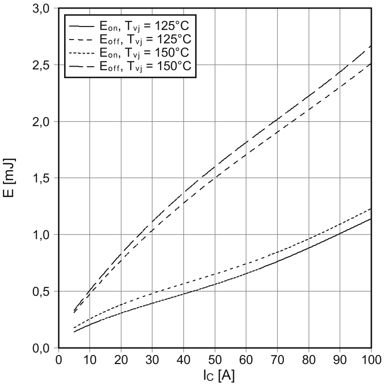
Ausgangskennlinienfeld IGBT-Wechselr. (typisch)
output characteristic IGBT-inverter (typical)
$I_C = f(V_{CE})$
$T_{vj} = 150$°C



Übertragungscharakteristik IGBT-Wechselr. (typisch)
transfer characteristic IGBT-inverter (typical)
$I_C = f(V_{GE})$
$V_{CE} = 20$ V



Schaltverluste IGBT-Wechselr. (typisch)
switching losses IGBT-inverter (typical)
$E_{on} = f(I_C)$, $E_{off} = f(I_C)$
$V_{GE} = \pm 15$ V, $R_{Gon} = 8.2\ \Omega$, $R_{Goff} = 8.2\ \Omega$, $V_{CE} = 300$ V



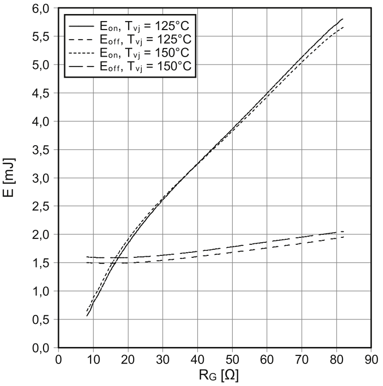
prepared by: DK	date of publication: 2010-01-06
approved by: MB	revision: 3.0

Technische Information / technical information

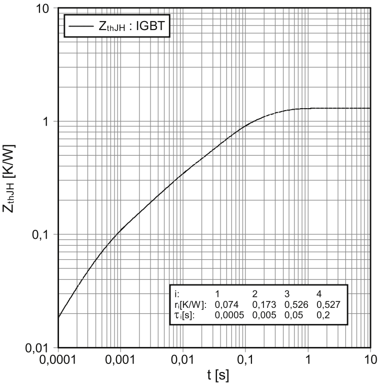
IGBT-Module
IGBT-modules

F4-50R06W1E3

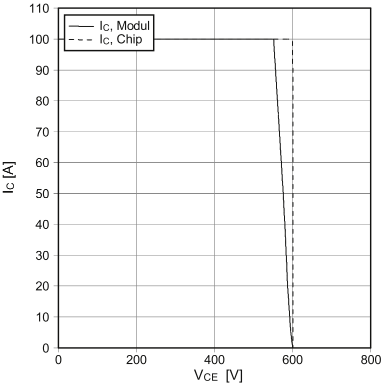
Schaltverluste IGBT-Wechselr. (typisch)
switching losses IGBT-Inverter (typical)
E_{on} = f (R_G), E_{off} = f (R_G)
V_{GE} = ±15 V, I_C = 50 A, V_{CE} = 300 V



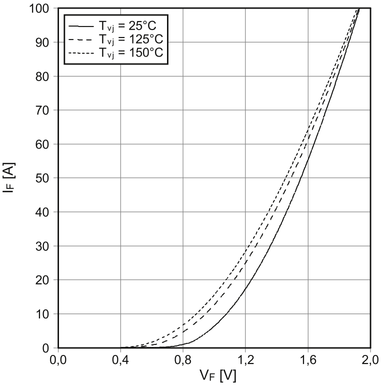
Transienter Wärmewiderstand IGBT-Wechselr.
transient thermal impedance IGBT-inverter
Z_{thJH} = f (t)



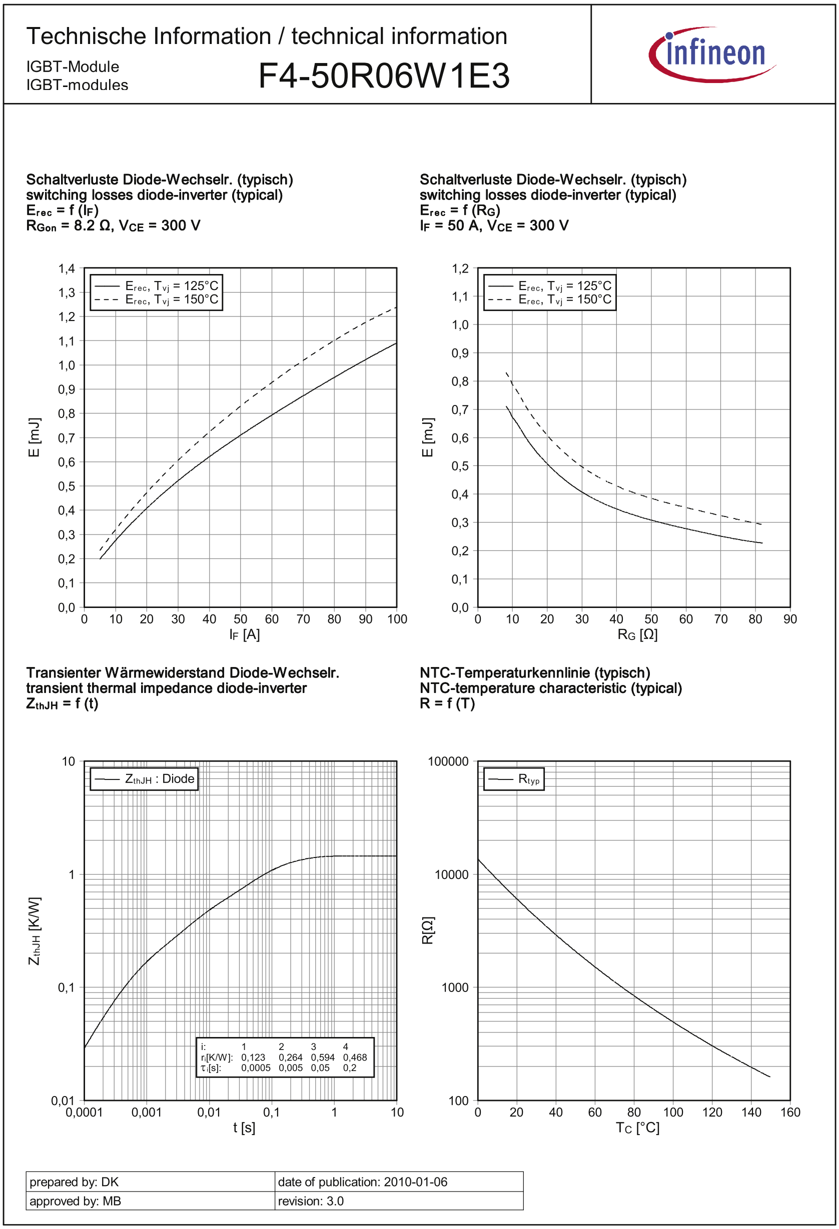
Sicherer Rückwärts-Arbeitsbereich IGBT-Wr. (RBSOA)
reverse bias safe operating area IGBT-inv. (RBSOA)
I_C = f (V_{CE})
V_{GE} = ±15 V, R_{Goff} = 8.2 Ω, T_{vj} = 150°C



Durchlasskennlinie der Diode-Wechselr. (typisch)
forward characteristic of diode-inverter (typical)
I_F = f (V_F)



prepared by: DK	date of publication: 2010-01-06
approved by: MB	revision: 3.0



Technische Information / technical information

IGBT-Module
IGBT-modules

F4-50R06W1E3

infineon

Schaltverluste Diode-Wechselr. (typisch)
switching losses diode-inverter (typical)
$E_{rec} = f(I_F)$
$R_{Gon} = 8.2\ \Omega$, $V_{CE} = 300$ V

Schaltverluste Diode-Wechselr. (typisch)
switching losses diode-inverter (typical)
$E_{rec} = f(R_G)$
$I_F = 50$ A, $V_{CE} = 300$ V

Transienter Wärmewiderstand Diode-Wechselr.
transient thermal impedance diode-inverter
$Z_{thJH} = f(t)$

ZthJH : Diode
ZthJH [K/W]
10
1
0,1
0,01
0,0001
0,001
0,01
0,1
1
10
t [s]

i:	1	2	3	4
r_i[K/W]:	0,123	0,264	0,594	0,468
τ_i[s]:	0,0005	0,005	0,05	0,2

NTC-Temperaturkennlinie (typisch)
NTC-temperature characteristic (typical)
R = f (T)

Rtyp
R[Ω]
100000
10000
1000
100
0
20
40
60
80
100
120
140
160
TC [°C]

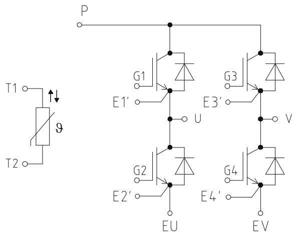
prepared by: DK	date of publication: 2010-01-06
approved by: MB	revision: 3.0

Technische Information / technical information

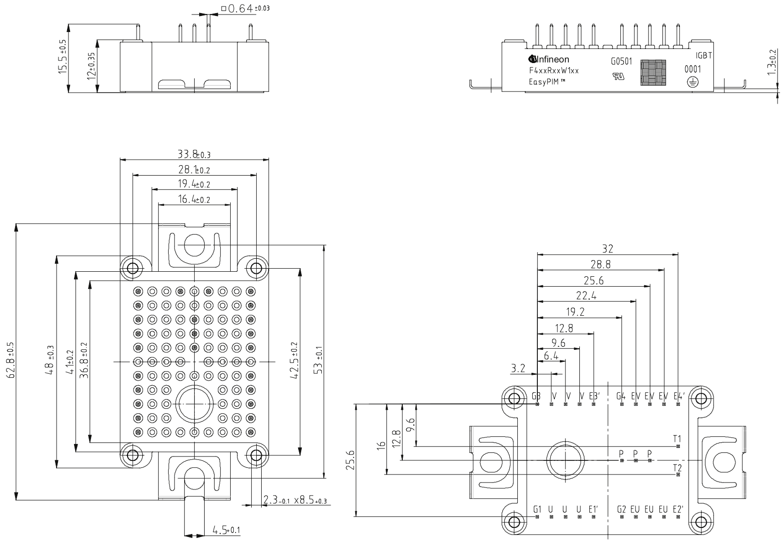
IGBT-Module
IGBT-modules

F4-50R06W1E3

Schaltplan / circuit diagram

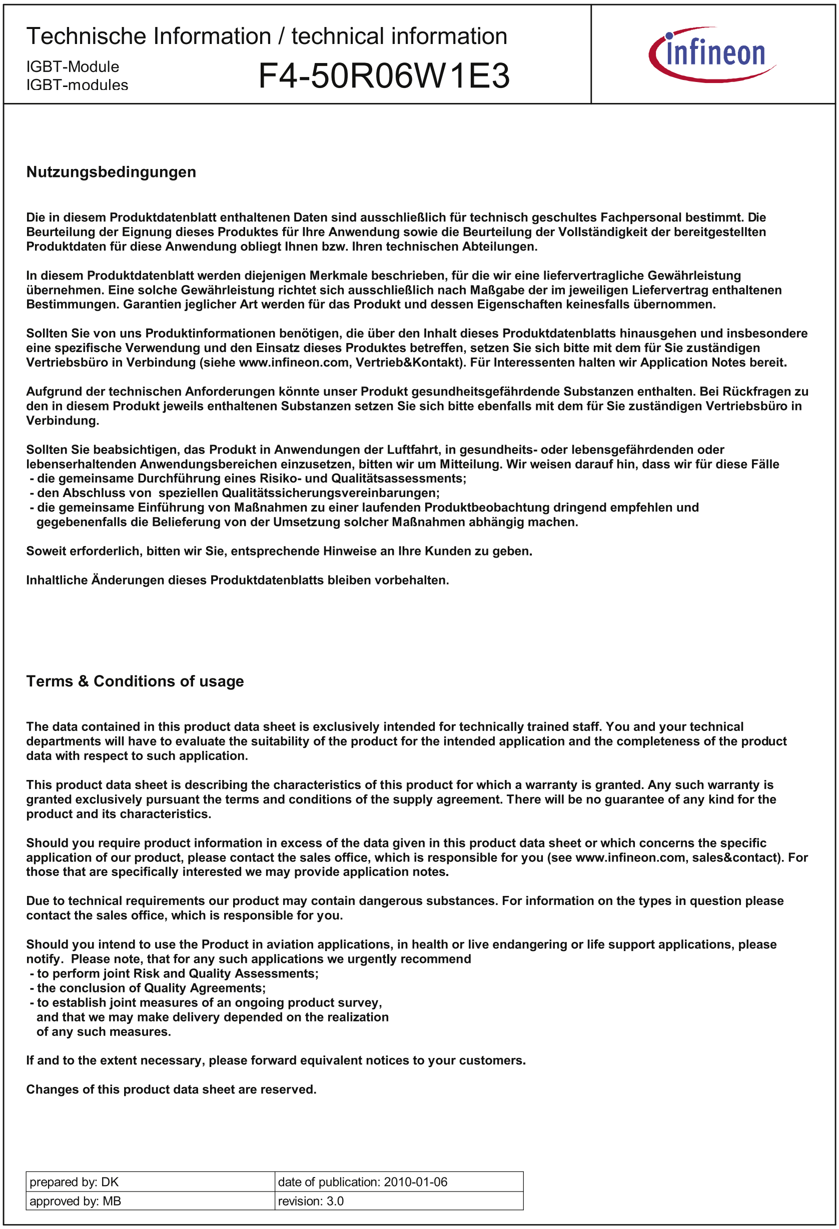

P
T1
T2
ϑ
G1
E1'
G3
E3'
U
V
G2
E2'
G4
E4'
EU
EV

Gehäuseabmessungen / package outlines


□0.64 ±0.03
15.5±0.5
12±0.35
Infineon
F4xxRxxW1xx
EasyPIM™
G0501
IGBT
0001
1.3±0.2
33.8±0.3
28.1±0.2
19.4±0.2
16.4±0.2
62.8±0.5
48±0.3
41±0.2
36.8±0.2
42.5±0.2
53±0.1
2.3-0.1
x8.5+0.3
4.5+0.1
32
28.8
25.6
22.4
19.2
12.8
9.6
6.4
3.2
25.6
16
12.8
9.6
G3 V V V E3'
G4 EV EV EV EV E4'
T1
P P P
T2
G1 U U U E1'
G2 EU EU EU E2'

Pinpositions with tolerance ⌖ ø0.4

prepared by: DK	date of publication: 2010-01-06
approved by: MB	revision: 3.0



Technische Information / technical information

IGBT-Module
IGBT-modules

F4-50R06W1E3

Nutzungsbedingungen

Die in diesem Produktdatenblatt enthaltenen Daten sind ausschließlich für technisch geschultes Fachpersonal bestimmt. Die Beurteilung der Eignung dieses Produktes für Ihre Anwendung sowie die Beurteilung der Vollständigkeit der bereitgestellten Produktdaten für diese Anwendung obliegt Ihnen bzw. Ihren technischen Abteilungen.

In diesem Produktdatenblatt werden diejenigen Merkmale beschrieben, für die wir eine liefervertragliche Gewährleistung übernehmen. Eine solche Gewährleistung richtet sich ausschließlich nach Maßgabe der im jeweiligen Liefervertrag enthaltenen Bestimmungen. Garantien jeglicher Art werden für das Produkt und dessen Eigenschaften keinesfalls übernommen.

Sollten Sie von uns Produktinformationen benötigen, die über den Inhalt dieses Produktdatenblatts hinausgehen und insbesondere eine spezifische Verwendung und den Einsatz dieses Produktes betreffen, setzen Sie sich bitte mit dem für Sie zuständigen Vertriebsbüro in Verbindung (siehe www.infineon.com, Vertrieb&Kontakt). Für Interessenten halten wir Application Notes bereit.

Aufgrund der technischen Anforderungen könnte unser Produkt gesundheitsgefährdende Substanzen enthalten. Bei Rückfragen zu den in diesem Produkt jeweils enthaltenen Substanzen setzen Sie sich bitte ebenfalls mit dem für Sie zuständigen Vertriebsbüro in Verbindung.

Sollten Sie beabsichtigen, das Produkt in Anwendungen der Luftfahrt, in gesundheits- oder lebensgefährdenden oder lebenserhaltenden Anwendungsbereichen einzusetzen, bitten wir um Mitteilung. Wir weisen darauf hin, dass wir für diese Fälle
- **die gemeinsame Durchführung eines Risiko- und Qualitätsassessments;**
- **den Abschluss von speziellen Qualitätssicherungsvereinbarungen;**
- **die gemeinsame Einführung von Maßnahmen zu einer laufenden Produktbeobachtung dringend empfehlen und gegebenenfalls die Belieferung von der Umsetzung solcher Maßnahmen abhängig machen.**

Soweit erforderlich, bitten wir Sie, entsprechende Hinweise an Ihre Kunden zu geben.

Inhaltliche Änderungen dieses Produktdatenblatts bleiben vorbehalten.

Terms & Conditions of usage

The data contained in this product data sheet is exclusively intended for technically trained staff. You and your technical departments will have to evaluate the suitability of the product for the intended application and the completeness of the product data with respect to such application.

This product data sheet is describing the characteristics of this product for which a warranty is granted. Any such warranty is granted exclusively pursuant the terms and conditions of the supply agreement. There will be no guarantee of any kind for the product and its characteristics.

Should you require product information in excess of the data given in this product data sheet or which concerns the specific application of our product, please contact the sales office, which is responsible for you (see www.infineon.com, sales&contact). For those that are specifically interested we may provide application notes.

Due to technical requirements our product may contain dangerous substances. For information on the types in question please contact the sales office, which is responsible for you.

Should you intend to use the Product in aviation applications, in health or live endangering or life support applications, please notify. Please note, that for any such applications we urgently recommend
- **to perform joint Risk and Quality Assessments;**
- **the conclusion of Quality Agreements;**
- **to establish joint measures of an ongoing product survey, and that we may make delivery depended on the realization of any such measures.**

If and to the extent necessary, please forward equivalent notices to your customers.

Changes of this product data sheet are reserved.

prepared by: DK	date of publication: 2010-01-06
approved by: MB	revision: 3.0

Index

Note: Page numbers followed by *f* indicate figures and *t* indicate tables.

A

Additive uncertainty model, 384–394, 384*t*, 385–386*f*, 388–389*f*
Air gap block, 405
Ampere-second balance, 440
Amplitude box, 374*f*
Analysis tools, 374, 374*f*
Apparent power *(S)*, 432–433
Audio susceptibility, zeta converter, 380–381, 380–381*f*
Average current drawn, calculation of, 449
Averaged equations, linearization of, 465–470
Average power, 424–425
  of linear circuits, 437–438
  RMS of signals, measurement of, 431–432, 432*f*
  signal, effective value of, 425–432
  sum of two periodic signals, effective value of, 429–431
Averaging, metal-oxide-semiconductor field-effect transistor, 462–465

B

Bipolar modulation, 301, 301*f*
Body diode
  modeling losses of, 328–331, 328–329*f*
    calculation of, 342, 344*f*
    conduction losses, 331, 333*f*
    turn-off losses, 330–331, 331–332*f*
    turn-on losses, 330
  thermal impedance, 331–334, 334*f*
Boost converter, 152*f*, 454–455, 454*f*
  calculated efficiency, 163*f*
  circuit simulation, ready-to-use modulator, 157–158
  diode settings, 153*f*
  dynamics of, 495–501, 495–496*f*, 497*t*, 501–502*f*
  efficiency measurement, 159–163, 159*f*
  generated error message, 164*f*
  inductor current waveform, 155*f*
  load voltage waveform, 155–156*f*
  metal-oxide-semiconductor field-effect transistor settings, 153*f*
  “Moving Average” block, 160–161*f*
  “Product” and “Divide” blocks, 160*f*
  pulse generator block, 152*f*
  pulse generator settings, 154*f*
  “Saturation” block settings, 159*f*, 161–162*f*
  Sawtooth PWM block, 157–158*f*
  simulation parameters, 154*f*, 162*f*
Buck-boost converter, 455, 455*f*
  dynamic of, 488–494, 488–489*f*, 490*t*, 494–495*f*
Buck converter, 446–452, 447*f*
  dynamical model of, 459–474, 460*f*, 474*f*
  output voltage of, 452–454, 452*f*
    boost converter, 454–455, 454*f*
    buck-boost converter, 455, 455*f*
    Cuk converter, 455–456, 456*f*
    DC-DC converters, types of, 454–457
    flyback converter, 456–457
  small signal transfer functions
    Analysis Tools window, 173*f*
    Analysis tools window, 169–170*f*
    Curve Properties window, 176*f*
    diode settings, 166*f*
    Edit curve properties, 176*f*
    Impulse Response Analysis, 170*f*
    inductor current transfer function, 178
    load resistance, 172–173*f*
    load trace data, 174*f*
    metal-oxide-semiconductor field-effect transistor settings, 165*f*
    output voltage waveform, 167*f*
    “Perturbation” and “Response” blocks settings, 167–168*f*
    “Response” block settings, 169*f*
    Sawtooth PWM block settings, 165*f*
    simulation parameters, 166*f*
    simulation preparation, 164–171
    simulation results, 172–177, 174*f*
    simulation results into MATLAB, 178–179
    Small Signal Analysis section, 168*f*
    voltage transfer function, 177

C

Cascaded inverter simulation, 252*f*
  configuration parameters window, 258–259, 259*f*
  lower inverter switches, control signals, 250, 251*t*
  metal-oxide-semiconductor field-effect transistor S1 settings, 252, 253*f*
  output voltage, 260, 260*f*
  Repeating Sequence block, 253, 254–256*f*, 255
  resistor R1 settings, 252, 253*f*
  sample waveform, 254, 254*f*
  simulation result (load voltage), 259, 259*f*
  Simulink model, 251, 251*f*
  single-phase inverter circuit, 247, 248–249*f*, 249
  stages, 260, 260*f*
  time domain equivalent of waveform, 254–255, 255*f*
  upper inverter switches, control signals, 250, 250*t*
Circuit simulation with piecewise linear electrical circuit simulation
  RC circuit analysis, 43–73
  resistive voltage divider, 5–43
Closed-loop buck converter, 277
Collector-emitter voltage, 321, 321*f*
Complex Power, 433
Conduction losses
  of body diode, 331, 333*f*
  for insulated gate bipolar transistor, 320–321, 320–321*f*
Constant Temperature block, 336–337*f*
Continuous current mode (CCM), 445
Controlled half-wave rectifier, 444–445, 444*f*
Controller designing, 276–283
Control signals, cascaded inverter
  for lower inverter switches, 250, 251*t*
  for upper inverter switches, 250, 250*t*
Control-to-inductor current transfer function, 284
Control-to-output transfer function, 500, 501*f*
  zeta converter, 511*f*, 515*f*
Cuk converter, 455–456, 456*f*
Current Mode Control (CMC), 475

D

DC-DC converter, 445–452, 446*f*
  average current drawn calculation, 449
  buck converter, 446–452, 447*f*
    output voltage, 454–457
  converter operating mode determination, 449–450, 449*f*
  dynamics of, 457–474
  frequency response extraction, 265–273
  output ripple, calculation of, 450–452, 451*f*
  output voltage, calculation of, 447–449
  uncertain model of, 383–396
    additive uncertainty model, 384–394, 384*t*, 385–386*f*, 388–389*f*
    multiplicative uncertainty model, 394–396, 395–396*f*
Diode block, 308*f*
Diode-clamped inverter simulation
  Library Browser window, 222, 222–223*f*
  PLECS blockset, 220, 220*f*, 223, 223*f*
  PLECS circuit block, 220, 221*f*
  Simulink environment, 219, 220*f*
Diode-clamped multilevel inverters
  simulation, 224*f*
    carriers, PWM signal, 226, 226*f*
    circuit simulation, 240–241
    line-line voltage, 241, 241*f*
    power stage, 225, 225*f*
    power stage components, 226–233
    PWM generation, 225, 225*f*, 234–239
    PWM signal, three-phase diode-clamped multilevel inverter, 226, 226*f*
  zeta converter, 519–520, 519–521*f*
Discontinuous conduction mode (DCM), 445
  buck converter, output voltage calculation, 452–454, 452*f*
    boost converter, 454–455, 454*f*
    buck-boost converter, 455, 455*f*
    Cuk converter, 455–456, 456*f*
    DC-DC converters, types of, 454–457
    flyback converter, 456–457

E

Electrical machine simulation
  armature current graph, 217*f*
  DC machine block, 211*f*
  DC Machine settings, 213*f*
  demultiplexer and scope blocks, 216*f*
  probe block, monitoring, 215–217, 215*f*
  Rotational Reference block, 213*f*
  rotational speed sensor block, 212*f*
  Rotor speed graph, 215*f*
  shaft speed graph, 217*f*
  signal demultiplexer block, 216*f*
  simulation parameter, 214*f*
  torque (Controlled) block, 212*f*
  Torque block settings, 214*f*
  variable load, 211*f*

F

Feedback control, on output impedance, 486–488, 486–488*f*
First-order Cauer thermal impedance network, 323*f*
First-order resistor–capacitor circuit
  AC voltage source block, 53–54*f*
  capacitor block, 45*f*
  capacitor initial voltage, 50–51*f*
  capacitor settings, 46*f*
  capacitor voltage, 49*f*
    waveform, 48*f*, 51*f*
  circuit current, 50*f*
    waveform, 48*f*
  circuit differential equation, 47
  closed-loop system, 68*f*, 70*f*
  continuous time blocks, 67*f*
  control systems simulation, 66–71
  DC voltage source settings, 45*f*
  discrete time blocks, 67*f*
  Green signals (control/measurement signals), 47*f*
  inductor initial current, 52*f*
  multiple input scope, 54–58
  multiplexer block, 51–52
  piecewise linear electrical circuit simulation, help system, 72–73
  Plant block settings, 70*f*
  RC circuit with AC source, 53*f*
  Ready-to-use simulation schematic, 71*f*
  resistor settings, 46*f*
  scope block, 59–66
  simulation parameters, 43–44, 47*f*
  simulation settings, 47*f*
  Steady-state Lissajous curve, 65*f*
  Step block, 69*f*
  summation block, 69*f*
  transfer function block, 68*f*
  voltmeter blocks, 52, 54*f*
  waveforms, same axis, 50–53
  XY Plot block, 59–66
Flux Rate Meter measurement, 415, 416*f*
Flyback converter, 456–457
Foster coefficients, 322
Fourier series, 434–438
  average power calculation using, 437–438
  of waveshapes, 435–436
Fourier series block, 204–207*f*
  Clock block, 208*f*
  controllable voltage source, 208*f*
  Gain block settings, 207*f*
  Voltage Source (Controlled) block, 209*f*
Frequency sweep range, 269
Full-Bridge Inverter block, 340, 340*f*
Full-Bridge Inverter subsystem, 303, 304*f*, 335
Function block, 305–306*f*

G

Graphs reading, 537–543*f*, 538, 540, 542–543

H

*H*α control, 368–369
H bridge inverters, zeta converter, 518, 518–519*f*
Heat Flow Meter block, 354*f*
Heat sink, 521–526, 522–525*f*
  designing, 352–357, 354–355*f*
  Heat Sink block, 335*f*
Hysteretic core block, 404, 404*f*

I

Impulse response analysis, 269, 269*f*
Inductance box, 385–386*f*
Input impedance, of converter, 478–486, 478–479*f*, 480*t*, 484–486*f*
Instantaneous power, 422–423, 422–423*f*
Insulated gate bipolar transistor (IGBT)
  conduction losses for, 320–321, 320–321*f*
  with diode block, 306, 306–307*f*
  switching losses
    calculation, 337–342, 339–344*f*
    turn-off, 317–319, 318–319*f*
    turn-on, 312–316, 312–317*f*
  thermal impedance, 321, 322*f*
  total losses, calculation of, 342–347, 345–347*f*
Inverter output voltage scope waveform, 310*f*
Inverters, zeta converter, 515–520, 515–516*f*

J

Junction temperatures measurement, 348, 348–349*f*

K

Kharitonov’s theorem, 368

L

Leakage flux path block, 405
Linear circuits
  power computations for, 433–434
    average power, calculation of, 437–438

Linear circuits *(Continued)*
    Fourier series, 434–438
    waveshapes, Fourier series of, 435–436
Linear Core block, 402

M

Magnetic blocks, 399–405, 400–401*f*
  air gap block, 405
  hysteretic core block, 404, 404*f*
  implementation, 405–406, 405–406*f*
  leakage flux path block, 405
  magnetic permeance block, 402
  saturable core block, 402–403, 403–404*f*
  winding block, 400–402, 401–402*f*
Magnetic circuits, 399
  electrical equivalent circuit, 408, 408*f*
  ferromagnetic core, 407–408, 408*f*
  flux density, 413, 413*f*
  Flux Rate Meter measurement, 415, 416*f*
  inductance, 409
  input DC source, addition of, 409, 411*f*
  Linear Core and Air Gap blocks addition, 409, 410*f*
  magnetic configuration, 406–407, 406–407*f*
  magnetomotive forces measurement, 415, 415–416*f*
  MMF distribution, 409
  Probe and Scope blocks, addition of, 409, 412*f*
  Probe Editor block, 410, 412*f*
  resistor current, 414, 414*f*
Magnetic permeance block, 402
Magnetic section of Library Browser, 399, 400*f*
Magnetomotive forces, magnetic circuits, 415, 415–416*f*
MATLAB® environment, 388, 392
  mutual inductance, 181*f*
  small-signal transfer functions using, 471–474, 473*f*
  total harmonic distortion (THD), 198*f*
Metal–oxide–semiconductor field-effect transistor (MOSFET), 524*f*
  with resistive load, 76*f*, 441–443, 441*f*
    average and RMS of waveform measurement, 82–85
    averaged equations, linearization of, 465–470
    averaging, 462–465
    buck converter, dynamical model of, 459–474, 460*f*
    buck converter, output voltage calculation, 452–454, 452*f*
    closed MOSFET, dynamical equations, 461
    comparator block, 79*f*
    constant block, 78*f*
    controlled half-wave rectifier, 444–445, 444*f*
    DC-DC converters, 445–452, 446*f*
    dissipated power, probe block, 86, 86–87*f*
    dynamics of DC-DC converters, 457–474
    input power waveform, 97–101, 98–100*f*
    load resistor average value, 85*f*
    load resistor voltage, 82*f*
    MATLAB®, small-signal transfer functions using, 471–474, 473*f*
    mean value measurement, 84*f*
    MOSFET block, 77*f*
    MOSFET gate signals, 78*f*, 82*f*
    on-resistance, 77*f*
    opened, dynamical equations, 461–462
    probe block, 86–88*f*
    PWM generation blocks, 78*f*, 91, 91*f*, 94–95*f*, 97*f*
    PWM signal using ready-to-use blocks, 101–103, 101–102*f*
    resistor power waveform, 88, 88–89*f*
    RMS, load resistor voltage, 85*f*
    RMS measurement, 85*f*
    Sawtooth PWM block, 103, 103–104*f*
    “Signal Inport” and “Signal Outport” blocks, 95, 96*f*
    simulation parameters, 81*f*
    simulation preparation, 76–82
    simulation result (“Vout” scope), 81*f*
    Space Vector PWM modulator block documentation, 101, 102*f*
    state space averaging, 457–459, 457*t*
    subsystem block, 89–97, 90–94*f*
    triangular wave generator block, 79–80*f*
    uncontrolled half-wave rectifier, 443–444, 443*f*
Model initialization commands, 389*f*
Modulation index, 301–302
Multiplicative uncertainty model, 394–396, 395–396*f*
μ synthesis, 369
Mutual inductance
  basic circuit theory, 180
  MATLAB codes, 181*f*
  mutual inductor block, 182*f*, 183, 184*f*
  parametric variables, component values, 186–188
  primary winding current, 185*f*
  secondary winding current, 185*f*

- simple circuit, 180*f*
- simulation parameters, 184*f*
- simulation preparation, 180–185
- time domain graph, 180, 181*f*
- transformers, 185, 185*f*

O

Open-loop voltage control function block, 303*f*
Output impedance
- of converter, 478–486, 478–479*f*, 480*t*, 484–486*f*
- extraction, 285–289
- feedback control on, 486–488, 486–488*f*
- zeta converter, 381–382, 382–383*f*

Output ripple, calculation of, 450–452, 451*f*
Output voltage
- of buck converter, 452–454, 452*f*
  - boost converter, 454–455, 454*f*
  - buck-boost converter, 455, 455*f*
  - Cuk converter, 455–456, 456*f*
  - DC-DC converters, types of, 454–457
  - flyback converter, 456–457
- calculation of, 447–449

P

Parametric uncertainty, 366, 366*f*
Periodic Average, 341, 342*f*
Periodic Impulse Average block, 341, 343–344*f*
Phase full-wave controlled rectifier
- controlled single-phase rectifiers, 150*f*
- controlled 3-phase rectifier, 142*f*
- delay block, 150–151, 150–151*f*
- "Electrical Label" blocks, 145–150, 146–147*f*
- input AC source block settings, 144*f*
- load current, 145*f*
- load voltage, 145*f*
- 6-pulse generator block, 143*f*
- "Signal From" and "Signal Goto" block, 145–150, 146*f*, 149*f*
- "Signal Goto," first output of multiplexer, 148, 148*f*
- simulation preparation, 142–145

Phase inverter
- comparator block, 189*f*
- Fourier series block, 204–208
- insulated gate bipolar transistor Half-Bridge block, 195*f*
- line-to-line voltage, 196*f*
- load, 195–196, 195*f*
- logical operator block, 190–192*f*, 194*f*
- phase A current, 196*f*
- phase A-ground voltage, 196*f*
- power stage, 194, 194–195*f*
- PWM generator, 188–193, 189*f*
- simulation preparation, 188–196
- sine wave generator block, 190*f*, 193*f*
- total harmonic distortion (THD), 197–203

Piecewise linear electrical circuit simulation (PLECS), 1
- magnetic circuits, 417, 417*f*, 419*f*
- magnetic configuration, 407*f*
- power electronic circuits simulation (*see* Power electronic circuits simulation with piecewise linear electrical circuit simulation)
- thermal search path of, 326–328, 326–328*f*

Power electronic circuits simulation with piecewise linear electrical circuit simulation
- boost converter, 152–163
- metal-oxide-semiconductor field-effect transistor with resistive load, 76–103
- mutual inductance, 180–188
- phase full-wave controlled rectifier, 142–151
- phase inverter, 188–208
- simulation of electrical machines, 209–217
- single-phase full-wave controlled rectifier, 131–141
- single-phase half-wave controlled rectifier, 127–131
- small signal transfer functions for buck converter, 164–179
- uncontrolled single-phase half-wave rectifier, 104–126

Power electronics converters, 38
- in piecewise linear electrical circuit simulation, 3

Power factor, 432–433
Power stage, diode-clamped multilevel inverter simulation, 225, 225*f*
- capacitor C1 settings, 226, 227*f*
- components, 226–233
- diode D1 settings, 226, 228*f*
- inductor L1 settings, 226, 229*f*
- insulated gate bipolar transistor, 226, 228*f*
- line-line voltage, 229
- resistor rC1 settings, 226, 227*f*
- resistor R1 settings, 226, 229*f*
- Signal From block, 230, 230–231*f*
- voltmeter block, 229, 230*f*

Power supply ripple rejection (PSRR), 380–381, 380–381*f*

Probe block, 261–264
Probe Editor block, 410, 412*f*
Proportional Integral Derivative (PID) controllers
  buck converter
    closed-loop control of, 475*f*
    dynamical model of, 474*f*
  PID controller block, 278, 278*f*
PWM scope waveforms, 308*f*, 310*f*

R

Resistive load
  metal-oxide-semiconductor field-effect transistor with, 441–443, 441*f*
    averaged equations, linearization of, 465–470
    averaging, 462–465
    buck converter, dynamical model of, 459–474, 460*f*
    buck converter, output voltage calculation, 452–454, 452*f*
    closed MOSFET, dynamical equations, 461
    controlled half-wave rectifier, 444–445, 444*f*
    DC-DC converters, 445–452, 446*f*
    dynamics of DC-DC converters, 457–474
    MATLAB®, small-signal transfer functions using, 471–474, 473*f*
    opened MOSFET, dynamical equations, 461–462
    state space averaging, 457–459, 457*t*
    uncontrolled half-wave rectifier, 443–444, 443*f*
Resistive voltage divider, 6*f*
  Adding text to schematic, 35–36
  ammeter block, 37–38, 40–41*f*
  axis limits, 22–23
  Bitmap Properties window, 34*f*
  block names, 35
  closed lock, 28*f*
  current scope waveform, 41–42*f*
  DC voltage source block, 8, 10*f*
  display block, 35, 36*f*
  drawn schematic export, 31–34
  Edit curve properties, 25*f*
  Electrical Ground block, 12, 18*f*
  Exported graphic file, 34*f*
  Hiding the block names, 35
  Library Browser window, 8, 10–11*f*
  Open lock, 28*f*
  output voltage measurement, 12, 16–17*f*
  PLECS Library Browser, 6, 7*f*
  PLECS start page, 6, 6*f*
  power circuit components, 42*f*
  Resistor block, 8, 11*f*
  resistor R2 voltage, 21, 21*f*
  Schematic editor window, 6, 8*f*
  scope block waveforms, 29–31
  Scope parameters, 21, 22*f*
  Simulation Parameter window, 20, 20*f*
  Voltmeter block, 12, 16–17*f*
  waveform properties, 24
  waveform values, 24–26
  wire colors, 38–43
  zoom icons, 29*f*
  zoom in/out, waveform, 26–28
Resistor–capacitor (RC) circuit analysis
  AC voltage source block, 53–54*f*
  capacitor block, 45*f*
  capacitor initial voltage, 50–51*f*
  capacitor settings, 46*f*
  capacitor voltage, 49*f*
    waveform, 48*f*, 51*f*
  circuit current, 50*f*
    waveform, 48*f*
  circuit differential equation, 47
  closed-loop system, 68*f*, 70*f*
  continuous time blocks, 67*f*
  control systems simulation, 66–71
  DC voltage source settings, 45*f*
  discrete time blocks, 67*f*
  Green signals (control/measurement signals), 47*f*
  inductor initial current, 52*f*
  multiple input scope, 54–58
  multiplexer block, 51–52
  piecewise linear electrical circuit simulation, help system, 72–73
  Plant block settings, 70*f*
  RC circuit with AC source, 53*f*
  Ready-to-use simulation schematic, 71*f*
  Resistor settings, 46*f*
  scope block, 59–66
  simulation parameters, 43–44, 47*f*
  simulation settings, 47*f*
  Steady-state Lissajous curve, 65*f*
  Step block, 69*f*
  Summation block, 69*f*
  transfer function block, 68*f*
  voltmeter blocks, 52, 54*f*

  waveforms, same axis, 50–53
  XY Plot block, 59–66
Robust control, 368–369
  *H*α control, 368–369
  Kharitonov's theorem, 368
  μ synthesis, 369
Root Mean Square (RMS), 426–427
  measurement, 431–432, 432*f*
Routh-Hurwitz stability test, 368

S

Saturable core block, 402–403, 403–404*f*
Sawtooth PWM block, 373, 374*f*
Semiconductor losses, 337
Semiconductor switches, thermal description, 311
Signal Multiplexer block, 304*f*
Simulation of electrical machines
  armature current graph, 217*f*
  DC machine block, 211*f*
  DC Machine settings, 213*f*
  demultiplexer and scope blocks, 216*f*
  probe block, monitoring, 215–217, 215*f*
  Rotational Reference block, 213*f*
  rotational speed sensor block, 212*f*
  Rotor speed graph, 215*f*
  shaft speed graph, 217*f*
  signal demultiplexer block, 216*f*
  simulation parameter, 214*f*
  torque (Controlled) block, 212*f*
  Torque block settings, 214*f*
  variable load, 211*f*
Simulation, running, 349–352, 349–352*f*
Simulink version of PLECS
  cascaded inverter, 247–260
  of cascaded inverter, 251, 251*f*
  circuit block, 243, 243*f*, 247, 247*f*
  Controlled voltage source, 244
  controller designing, 276–283
  control-to-inductor current transfer function, 284
  DC-DC converters, frequency response extraction, 265–273
  diode-clamped inverter, 219–223
  diode-clamped multilevel inverter, 224–241
  fitting a transfer function, 273–275
  output impedance extraction, 285–289
  probe block, 261–264
  Signal Inport block, 245, 245*f*, 247
  Signal Outport block, 242, 242*f*
  signals, sending/receiving, 241–247
  steady-state analysis, 289–294
Single-phase full-wave controlled rectifier
  discrete fourier block settings, 141*f*
  "Discrete Fourier Transform" block settings, 138–139*f*
  load voltage, 140–141*f*
Single-phase half-wave controlled rectifier
  controlled single-phase halfwave rectifier, 128, 128*f*
  load voltage, 131*f*
  pulse generator settings, 128, 129*f*
  simulation parameters, 130*f*
  simulation preparation, 128–130
  thyristor block, 128, 128*f*
  thyristor gate pulses, 130, 130*f*
  thyristor gate signal, 131*f*
Single-phase inverter, 249, 249*f*
  circuit, 247, 248*f*
Single-phase open loop inverter, 300–302, 300–301*f*, 301*t*
  electrical simulation of, 302–310, 302–307*f*, 309–311*f*
Small Signal Perturbation blocks, 371, 371*f*
Small Signal Response block, 371, 371*f*
Small signal transfer functions
  for buck converter
    Analysis Tools window, 169–170*f*, 173*f*
    Curve Properties window, 176*f*
    diode settings, 166*f*
    Edit curve properties, 176*f*
    Impulse Response Analysis, 170*f*
    inductor current transfer function, 178
    load resistance, 172–173*f*
    load trace data, 174*f*
    metal-oxide-semiconductor field-effect transistor settings, 165*f*
    output voltage waveform, 167*f*
    "Perturbation" and "Response" blocks settings, 167–168*f*
    "Response" block settings, 169*f*
    Sawtooth PWM block settings, 165*f*
    simulation parameters, 166*f*
    simulation preparation, 164–171
    simulation results comparison, 172–177, 174*f*
    simulation results into MATLAB, 178–179
    Small Signal Analysis section, 168*f*
    voltage transfer function, 177
  using MATLAB®, 471–474, 473*f*
State space averaging (SSA), 457–459, 457*t*

Steady-state analysis, 289–294, 355*f*
Structured uncertainty, 367–368
Switching losses, 311–319, 311*f*
  insulated gate bipolar transistor
    turn-off, 317–319, 318–319*f*
    turn-on, 312–316, 312–317*f*
Symmetrical PWM block, 303*f*
System Identification Toolbox, 273

T
Taylor series, 466
Thermal impedance, 321–324, 322–324*f*
  body diode, 331–334, 334*f*
Theta block, 302*f*
Total harmonic distortion (THD), 197–203, 200–203*f*, 438–439
  "Discrete Total Harmonic Distortion" block, 197*f*, 199*f*, 201*f*, 203*f*
  MATLAB code, 198*f*
  of output voltage, 249
Transfer function, 273–275

U
Uncertain model
  of DC-DC converters, 383–396
    additive uncertainty model, 384–394, 384*t*, 385–386*f*, 388–389*f*
    multiplicative uncertainty model, 394–396, 395–396*f*
Uncertainty, 365
  parametric uncertainty, 366, 366*f*
  structured uncertainty, 367–368
  unstructured uncertainty, 367
Uncontrolled half-wave rectifier, 443–444, 443*f*
Uncontrolled single-phase half-wave rectifier
  base frequency, 111*f*
  descriptive labels, 108*f*
  diode block, 105*f*
  diode settings, 106*f*
  "discrete mean value" block, 120–122, 120–122*f*
  "discrete RMS value" block, 116–119, 117–119*f*
  forward biased diode, 106*f*
  Fourier analysis, 109*f*
  Fourier spectrum, 109*f*
  fundamental frequency, 111*f*
  harmonic analysis, 109
  harmonic content of output, 107–111
  input AC source settings, 105*f*
  instantaneous load power waveform, 124–126, 124–127*f*
  "load voltage RMS" scope waveform, 119–120*f*
  maximum/minimum of waveforms, scope block, 122–123, 123–124*f*
  output voltage/current, 114–116
  RMS values of voltages/currents, 112–113
  simulation parameters, 106*f*
  simulation preparation, 104–107
  single-phase half-bridge rectifier, 104*f*
Unipolar PWM inverter, 358–359, 359*f*
  losses, calculation of, 360–363, 361–362*f*
Unstructured uncertainty, 367

V
Voltage Mode Control (VMC), 475
Voltage waveforms of cascaded inverter, 249, 250*f*
Volt-Second balance, 439–440, 448

W
WebPlotDigitizer, 537–538*f*
Winding block, 400–402, 401–402*f*

Z
Zeta converter, 369–396
  audio susceptibility, 380–381, 380–381*f*, 511*f*, 514*f*
  circuit differential equations, 504
  control-to-output transfer function, 511*f*, 515*f*
  diode-clamped multilevel inverters, 519–520, 519–521*f*
  dynamics of, 502–514, 503–504*f*, 503*t*, 510–512*f*, 514–517*f*, 519–520*f*, 522–525*f*
  electric circuit equivalent, 523–524*f*
  equivalent circuit, 503–504*f*
  H bridge inverters, 518, 518–519*f*
  heat sink equations, 522*f*
  inverters, 515–520, 515–516*f*
  output impedance, 381–382, 382–383*f*, 510*f*, 513*f*
  output voltage, 517*f*
  parameters, 384*t*, 503*t*
  RL type of loads, load current waveform for, 516*f*
  system analyzation without uncertainty, 370–379, 370*t*, 370–379*f*

SIMULATION OF POWER ELECTRONICS CONVERTERS USING PLECS®

FARZIN ASADI AND KEI EGUCHI

- Unique examples are included in the simulation of power electronics converters using PLECS®maybe say why PLECs is better/unique?
- Include explanations and guidance for all included simulations for redoing the simulations in own areas of application.
- Incorporates analysis and design for creating power electronics circuits rapidly with high accuracy.

Power electronic systems are widely used today to provide power processing for applications ranging from computing and communications, appliance control, transportation, and high-power transmission. *Simulation of Power Electronics Converters Using PLECS®* is a guide to simulating a power electronics circuit using the powerful new? Widely used? Most advanced? software for power electronics circuit simulation purposes.

Modeling and simulation are essential ingredients of the analysis and design process in power electronics. This book assists engineers to gain an increased understanding of circuit operation so that they can, for a given set of specifications, choose a topology, select appropriate circuit component types and values, estimate circuit performance, and complete the design by ensuring that the circuit performance will meet specifications even with the anticipated variations in operating conditions and circuit component values.

This book covers the fundamentals of power electronics converter simulation along with analysis of power electronics converters using PLECS® and concludes with real-world simulation examples for applied content making this book useful for all those in the electrical and electronics engineering field.

- Digital Power Electronics and Applications, Luo, 2005, 464p, 9780120887576
- Alternative Energy in Power Electronics, Rashid, 2014, 378p, 9780124167148
- Power Electronics Handbook 4ed, Rashid, 2017, 1522p, 9780128114070

Farzin Asadi, Professor in the Mechatronics Engineering Department at Kocaeli University in Kocaeli, Turkey. Mechatronics Engineering Department, Kocaeli University, Umuttepe Yerleşkesi 41380, Kocaeli, Turkey. Farzin Asadi, received his BSc degree in Electronics Engineering (Kordestan University, Kordestan, Iran), MSc degree in Control Engineering (Kordestan University, Kordestan, Iran) and PhD degree in Mechatronics Engineering (Kocaeli University, Kocaeli, Turkey). Currently, he is with the department of Mechatronics Engineering at the Kocaeli University, Kocaeli, Turkey. Farzin has published 30 international papers and ten books and is on the editorial board of six scientific journals. His research interests include switching converters, control theory, robust control of power electronics converters, and robotics.

Kei Eguchi, Professor at Fukuoka Institute of Technology, Fukuoka, Japan. Department of Information Electronics, Fukuoka Institute of Technology, 3-30-1 Wajirohigashi, Higashi-ku, Fukuoka, 811-0295 Japan. Kei Eguchi received the BEng, the MEng, and the DEng degrees from Kumamoto University, Kumamoto, Japan in 1994, 1996, and 1999, respectively. His research interests include nonlinear dynamical systems, intelligent circuits and systems, and low-voltage analog integrated circuits. From 1999 to 2006, he was an associate professor and a lecturer in Kumamoto National College of Technology. From 2006 to 2012, he was an associate professor in Shizuoka University. In 2012, he joined the faculty of Fukuoka Institute of Technology, where he is now a professor. In 2015, he was a visiting scholar, California State University, Eastbay.

ISBN 978-0-12-817364-0

ELSEVIER

ACADEMIC PRESS
An imprint of Elsevier
elsevier.com/books-and-journals

9 780128 173640